Contents

Chapter 1 Technical Overview 1-1

Introduction 1-2
About the Documentation 1-2

Applications 1-3
Heterogeneous Slope Overlying Bedrock 1-3
Block Failure Analysis 1-3
External Loads and Reinforcements 1-4
Complex Pore-Water Pressure Condition 1-5
Stability Analysis Using Finite Element Stress 1-5
Probabilistic Stability Analysis 1-6

Features and Capabilities 1-8
User Interface 1-8
Slope Stability Analysis 1-18

Using SLOPE/W 1-24
Defining Problems 1-24
Solving Problems 1-26
Contouring and Graphing Results 1-27

Formulation 1-28

Product Integration 1-30

Product Support 1-31

Chapter 2 Installing SLOPE/W 2-1

Basic Windows Skills 2-3
Windows Fundamentals 2-3
Locating Files and Directories 2-3
Viewing Data Files 2-4
System Messages 2-5

Basic SLOPE/W Skills 2-6
Starting and Quitting SLOPE/W 2-6
Dialog Boxes in SLOPE/W 2-6
Using Online Help 2-8

Installing the Software 2-9
Using the CD-ROM 2-9
Installing GEO-SLOPE Evaluation Software 2-9
Viewing GEO-SLOPE Manuals 2-9
Installing SLOPE/W 2-10

Installing Additional Software for Network Versions 2-16
The Rainbow NetSentinel Software 2-16

Network Version Requirements....2-17
Installing the Rainbow Network Software....2-17
Security Server Reference....2-18
Security Monitor Reference....2-36
NetSentinel Configuration Reference....2-41

Chapter 3 SLOPE/W Tutorial....3-1

An Example Problem....3-3

Defining the Problem....3-4
Set the Working Area....3-4
Set the Scale....3-5
Set the Grid Spacing....3-6
Saving the Problem....3-6
Sketch the Problem....3-7
Specify the Analysis Methods....3-9
Specify the Analysis Options....3-10
Define Soil Properties....3-10
Draw Lines....3-12
Draw Piezometric Lines....3-14
Draw the Slip Surface Radius....3-16
Draw the Slip Surface Grid....3-17
View Preferences....3-19
Sketch Axes....3-21
Display Soil Properties....3-23
Label the Soils....3-26
Add a Problem Identification Label....3-30
Verify the Problem....3-33
Save the Problem....3-35

Solving the Problem....3-36
Start Solving....3-36
Quit SOLVE....3-37

Viewing the Results....3-38
Draw Selected Slip Surfaces....3-39
View Method....3-40
View the Slice Forces....3-41
Draw the Contours....3-42
Draw the Contour Labels....3-43
Plot a Graph of the Results....3-44
Print the Drawing....3-47

Using Advanced Features of SLOPE/W....3-49
Specify a Rigorous Method of Analysis....3-49
Perform a Probabilistic Analysis....3-50
Import a Picture....3-60

Chapter 4 DEFINE Reference 4-1

Introduction **4-3**

Toolbars **4-4**
- Standard Toolbar 4-4
- Mode Toolbar 4-7
- View Preferences Toolbar 4-9
- Grid Toolbar 4-10
- Zoom Toolbar 4-11

The File Menu **4-13**
- File New 4-13
- File Open 4-15
- File Import: Data File 4-16
- File Import: Picture 4-17
- File Export 4-19
- File Save As 4-20
- File Print 4-22
- File Save Default Settings 4-23

The Edit Menu **4-25**
- Edit Copy All 4-25

The Set Menu **4-26**
- Set Page 4-26
- Set Scale 4-27
- Set Grid 4-29
- Set Zoom 4-30
- Set Axes 4-31

The View Menu **4-33**
- View Point Information 4-33
- View Soil Properties 4-34
- View Preferences 4-36
- View Toolbars 4-39
- View Redraw 4-40

The KeyIn Menu **4-41**
- KeyIn Project ID 4-42
- KeyIn Analysis Method 4-44
- KeyIn Analysis Control 4-53
- KeyIn Soil Properties 4-56
- KeyIn Strength Functions Shear/Normal 4-67
- KeyIn Strength Functions Anisotropic 4-79
- KeyIn Tension Crack 4-80
- KeyIn Points 4-83
- KeyIn Lines 4-85
- KeyIn Slip Surface Grid & Radius 4-89
- KeyIn Slip Surface Axis 4-93
- KeyIn Slip Surface Specified 4-94
- KeyIn Slip Surface Left Block 4-96
- KeyIn Slip Surface Right Block 4-99
- KeyIn Slip Surface Limits 4-101

KeyIn Pore Pressure: Water Pressure....4-102
KeyIn Pore Pressure: Air Pressure....4-109
KeyIn Load: Line Loads....4-110
KeyIn Load: Anchor Loads....4-111
KeyIn Load: Seismic Load....4-114
KeyIn Pressure Lines....4-115

The Draw Menu....4-118
Draw Points....4-119
Draw Points on Mesh....4-119
Draw Lines....4-120
Draw Slip Surface Grid....4-123
Draw Slip Surface Radius....4-126
Draw Slip Surface Axis....4-129
Draw Slip Surface Specified....4-130
Draw Slip Surface Left Block....4-132
Draw Slip Surface Right Block....4-137
Draw Slip Surface Limits....4-141
Draw Pore-Water Pressure....4-141
Draw Line Loads....4-146
Draw Anchor Loads....4-149
Draw Pressure Lines....4-152
Draw Tension Crack Line....4-155

The Sketch Menu....4-157
Sketch Lines....4-157
Sketch Circles....4-158
Sketch Arcs....4-158
Sketch Text....4-159
Sketch Axes....4-165

The Modify Menu....4-166
Modify Objects....4-166
Modify Text....4-169
Modify Pictures....4-171

The Tools Menu....4-176
Tools Verify....4-176
Tools SOLVE....4-179
Tools CONTOUR....4-179

The Help Menu....4-180

Chapter 5 SOLVE Reference....5-1

Introduction....5-2

The File Menu....5-3
File Open Data File....5-3

The Help Menu 5-6

Running SOLVE 5-7

Files Created for Limit Equilibrium Methods 5-10
Factor of Safety File - Limit Equilibrium Method 5-10
Slice Forces File - Limit Equilibrium Method 5-12
Probability File - Limit Equilibrium Method 5-15

Files Created for the Finite Element Method 5-17
Factor of Safety File - Finite Element Method 5-17
Slice Forces File - Finite Element Method 5-17
Probability File - Finite Element Method 5-20

Chapter 6 CONTOUR Reference 6-1

Introduction 6-3

Toolbars 6-4
Standard Toolbar 6-4
Mode Toolbar 6-5
View Preferences Toolbar 6-6
Method Toolbar 6-7

The File Menu 6-9
File Open 6-10

The Edit Menu 6-12

The Set Menu 6-13

The View Menu 6-14
View Method 6-14
View Slice Forces 6-15
View Preferences 6-18
View Toolbars 6-22

The Draw Menu 6-24
Draw Contours 6-24
Draw Contour Labels 6-25
Draw Slip Surfaces 6-26
Draw Graph 6-29
Draw Probability 6-36

The Sketch Menu....6-41
The Modify Menu....6-42
The Help Menu....6-43

Chapter 7 Modelling Guidelines....7-1
Introduction....7-3
Modelling Progression....7-4
Units....7-5
Selecting Appropriate X and Y Coordinates....7-6
Adopting a Method....7-7
Effect of Soil Properties on Critical Slip Surface....7-10
Steep Slip Surfaces....7-11
Weak Subsurface Layer....7-12
Seismic Loads....7-13
Geofabric Reinforcement....7-16
Structural Elements....7-17
Active and Passive Earth Pressures....7-18
Partial Submergence....7-20
Complete Submergence....7-21
Right-To-Left Analysis....7-22
Pore-Water Pressure Contours....7-23
Finite Element Stress Method....7-24
Probabilistic Analysis....7-25

Chapter 8 Theory....8-1
Introduction....8-3
Definition Of Variables....8-4
General Limit Equilibrium Method....8-9
Moment Equilibrium Factor Of Safety....8-10
Force Equilibrium Factor Of Safety....8-11

Slice Normal Force at the Base 8-12
Unrealistic m-alpha Values 8-13

Interslice Forces 8-16
Corps of Engineers Interslice Force Function 8-18
Lowe-Karafiath Interslice Force Function 8-19
Fredlund-Wilson-Fan Interslice Force Function 8-20

Effect Of Negative Pore-Water Pressures 8-23
Factor of Safety for Unsaturated Soil 8-23
Use of Unsaturated Shear Strength Parameters 8-24

Solving For The Factors Of Safety 8-25
Stage 1 Solution 8-25
Stage 2 Solution 8-25
Stage 3 Solution 8-26
Stage 4 Solution 8-27

Simulation of the Various Methods 8-30

Spline Interpolation of Pore-Water Pressures 8-34

Finite Element Pore-Water Pressure 8-36

Slice Width 8-37

Moment Axis 8-39

Soil Strength Models 8-41
Anisotropic Strength 8-41
Anisotropic Strength Modifier Function 8-42
Shear/Normal Strength Function 8-42

Finite Element Stress Method 8-44
Stability Factor 8-44
Normal Stress and Mobilized Shear Stress 8-45

Probabilistic Slope Stability Analysis 8-48
Monte Carlo Method 8-48
Parameter Variability 8-48
Normal Distribution Function 8-49
Random Number Generation 8-49
Estimation of Input Parameters 8-50
Correlation Coefficient 8-50
Statistical Analysis 8-51
Probability of Failure and Reliability Index 8-53
Number of Monte Carlo Trials 8-54

Chapter 9 Verification 9-1

Introduction 9-3

Comparison with Hand Calculations 9-4
Lambe and Whitman's Solution 9-4
SLOPE/W Solution Hand Calculated 9-6

Comparison with Stability Charts 9-10
Bishop and Morgenstern's Solution 9-10
SLOPE/W Solution Stability Chart 9-10

Comparison with Closed Form Solutions 9-12
Closed Form Solution for an Infinite Slope 9-12
SLOPE/W Solution Closed Form 9-14

Comparison Study 9-15

Illustrative Examples 9-17
Example with Circular Slip Surfaces 9-17
Example with Composite Slip Surfaces 9-17
Example with Fully Specified Slip Surfaces 9-18
Example with Block Slip Surfaces 9-19
Example with Pore-Water Pressure Data Points 9-20
Example with SEEP/W Pore-Water Pressure 9-21
Example with Slip Surface Projection 9-23
Example with Geofabric Reinforcement 9-23
Example with Anchors 9-25
Example with Finite Element Stresses 9-27
Example with Anisotropic Strength 9-29
Example with Probabilistic Analysis 9-33

References 10-1

Appendix A DEFINE Data File Description A-1

Introduction A-3

FILEINFO Keyword A-4

TITLE Keyword A-5

ANALYSIS Keyword A-6

CONVERGE Keyword A-8

SIDE Keyword A-9

LAMBDA Keyword A-10

SOIL Keyword A-11

SFUNCTION Keyword A-12

AFUNCTION Keyword A-13

POINT Keyword A-14

LINE Keyword A-15

TENSION Keyword A-16

GRID Keyword A-17

RADIUS Keyword A-18

AXIS Keyword A-19

LIMIT Keyword A-20

SLIP Keyword A-21

BLOCK Keyword A-22

PORU Keyword A-23

PIEZ Keyword A-24

PCON Keyword A-25

POGH Keyword A-26

POGP Keyword A-27

POGR Keyword A-28

PORA Keyword A-29

LOAD Keyword A-30

ANCHOR Keyword **A-31**

PBOUNDARY Keyword **A-32**

SEISMIC Keyword **A-33**

NODE Keyword **A-34**

ELEMENT Keyword **A-35**

MATLCOLOR Keyword **A-36**

Index

Chapter 1 Technical Overview

Introduction 1-3
About the Documentation 1-3

Applications 1-4
Heterogeneous Slope Overlying Bedrock 1-4
Block Failure Analysis 1-4
External Loads and Reinforcements 1-5
Complex Pore-Water Pressure Condition 1-6
Stability Analysis Using Finite Element Stress 1-6
Probabilistic Stability Analysis 1-7

Features and Capabilities 1-9
User Interface 1-9
Slope Stability Analysis 1-19

Using SLOPE/W 1-25
Defining Problems 1-25
Solving Problems 1-27
Contouring and Graphing Results 1-28

Formulation 1-29

Product Integration 1-31

Product Support 1-32

Introduction

SLOPE/W is a software product that uses limit equilibrium theory to compute the factor of safety of earth and rock slopes. The comprehensive formulation of SLOPE/W makes it possible to easily analyze both simple and complex slope stability problems using a variety of methods to calculate the factor of safety. SLOPE/W has application in the analysis and design for geotechnical, civil, and mining engineering projects.

SLOPE/W is a 32-bit, graphical software product that operates under Microsoft Windows 95 and Windows NT. The common "look and feel" of Windows applications makes it easy to learn how to use SLOPE/W, especially if you are already familiar with the Windows environment.

About the Documentation

The SLOPE/W documentation is divided into nine chapters and one appendix. Chapter 1 provides an overview of the product including its features and capabilities, how the product is used, and its formulation. Chapter 2 provides information on installing the software, including installation of the network version. Chapter 3 provides a step-by-step tutorial where a specific problem is defined, the solution computed, and the results viewed. Chapters 4, 5, and 6 contain detailed reference material for the DEFINE, SOLVE and CONTOUR programs. Chapter 7 gives guidelines for modelling many varied situations and is useful for finding practical solutions to modelling problems. Chapter 8 contains the details of the formulation including the alternative finite element stress approach and the implementation of the probabilistic stability analysis. In Chapter 9, model verification examples are presented to illustrate the correct numerical solution to problems for which an analytical solution exists. A series of example problems are also presented to illustrate the uses and capabilities of the software. The appendix presents the details of the data file format generated by the DEFINE program.

The Getting Started Guide contains only Chapters 1 through 3. The documentation in its entirety is available in the on-line help system and on the distribution CD-ROM as Microsoft Word document, (.DOC), files. You can use these files to print some or all of the documentation to meet your own requirements. If you do not have Microsoft Word, you can use the Word Viewer application included on the CD-ROM.

Applications

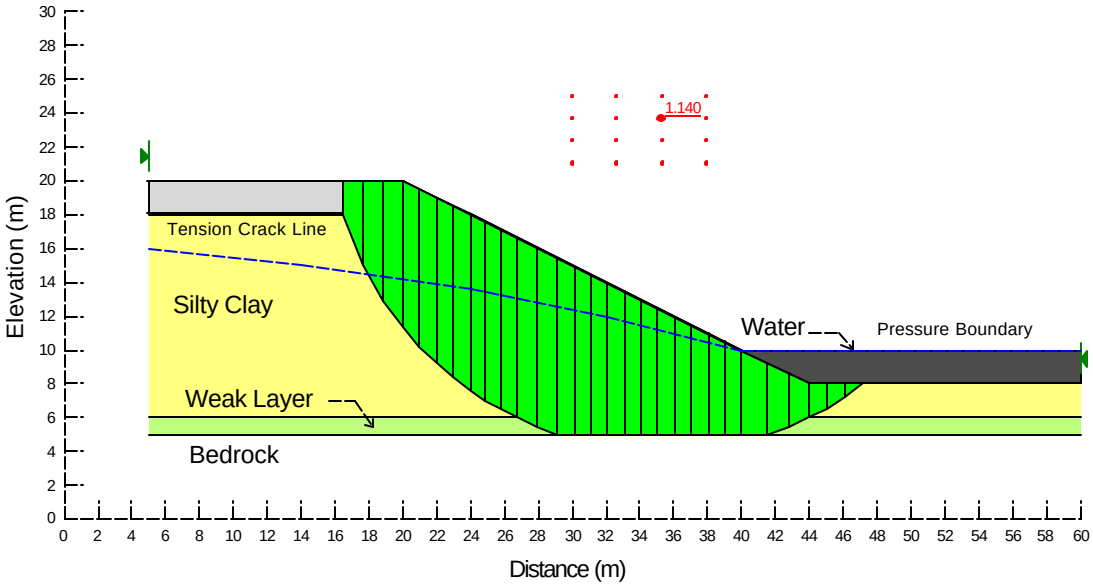
SLOPE/W is a powerful slope stability analysis program. Using limit equilibrium, it has the ability to model heterogeneous soil types, complex stratigraphic and slip surface geometry, and variable pore-water pressure conditions using a large selection of soil models. Analyses can be performed using deterministic or probabilistic input parameters. In addition, stresses computed using finite element stress analysis may be used in the limit equilibrium computations for the most complete slope stability analysis available. The combination of all these features means that SLOPE/W can be used to analyze almost any slope stability problem you will encounter.

This section gives a few examples of the many kinds of problems that can be modelled using SLOPE/W.

Heterogeneous Slope Overlying Bedrock

Figure 1.1 shows a typical slope stability problem. This specific case has a problematic weak layer located above impenetrable bedrock with a stronger silty clay layer above. The toe of the slope is beneath water, groundwater flows towards the toe, and a tension crack zone has developed at the crest of the slope. The slip surface for this slope is a composite circular arc with straight portions along the bedrock and in the tension crack zone. The Ordinary, Bishop, Janbu Simplified, Spencer, and Morgenstern-Price factors of safety can all be computed for this composite slip surface.

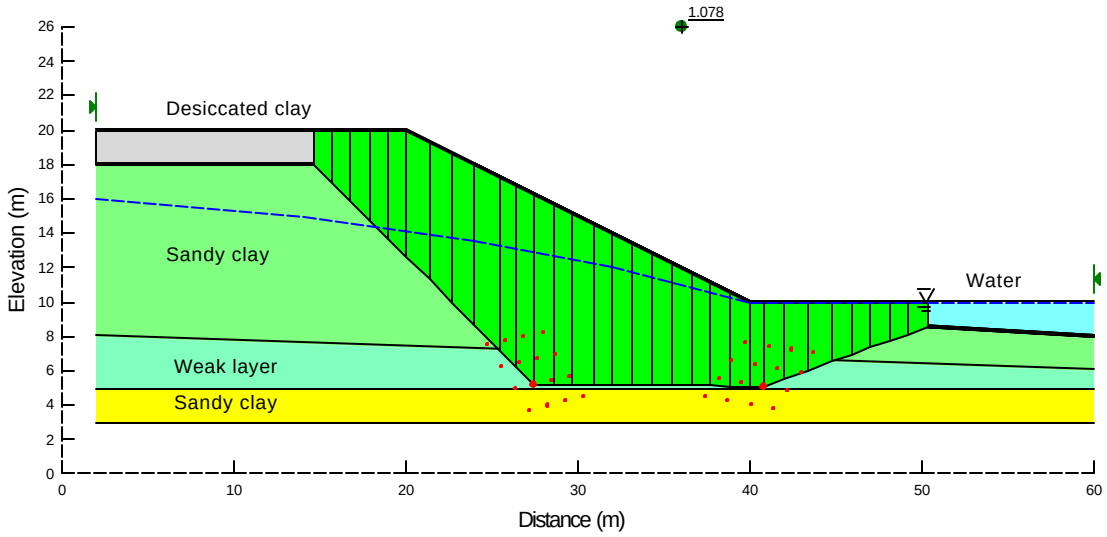
Figure 1.1 Heterogeneous Slope Overlying Bedrock



Block Failure Analysis

Figure 1.2 shows a slope stability analysis problem in a system of weak and strong stratigraphy. As shown in the figure, the analysis considers a block failure mode. This analysis also has the toe of the slope beneath water, groundwater flow towards the toe, and a tension crack zone at the crest. A large number of block slip surfaces can be analyzed by specifying a grid of points at the two lower corners of the block. The slip surface is projected upwards from these grid points at a user-specified range of angles.

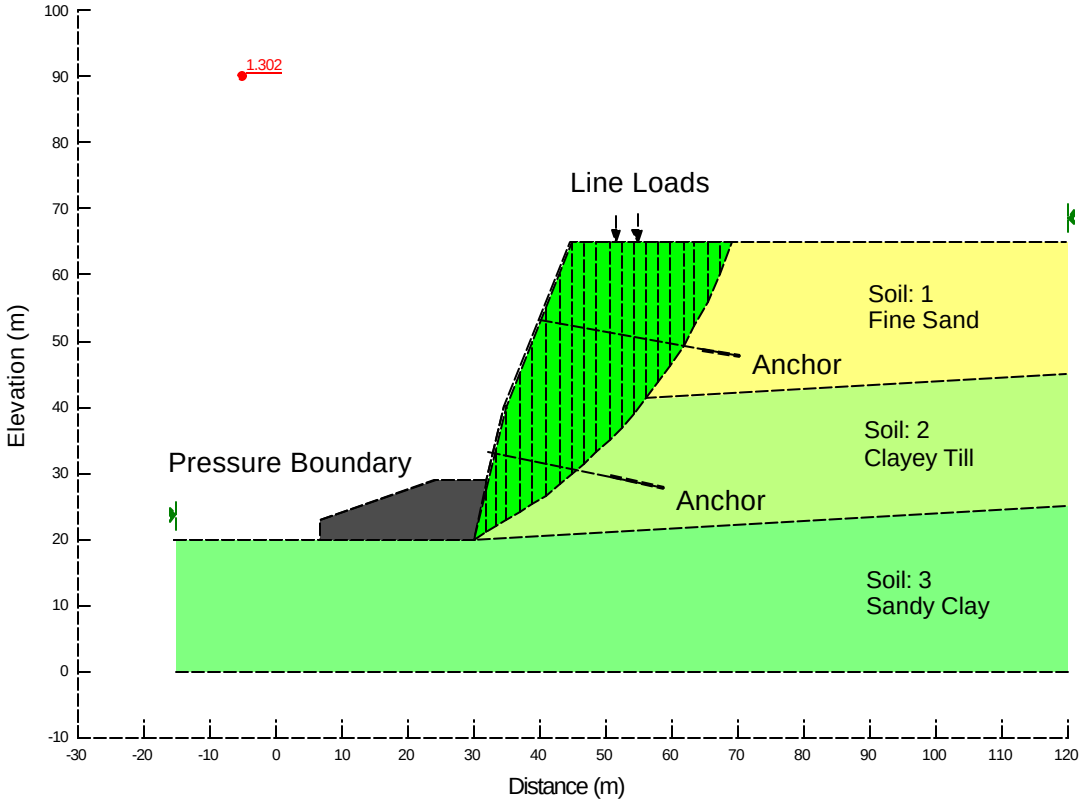
Figure 1.2 Block Failure Mode



External Loads and Reinforcements

SLOPE/W can calculate the factor of safety for slopes that are externally loaded and reinforced with anchors or geo-fabrics. Figure 1.3 shows the SLOPE/W analysis of a slope reinforced using anchors and subjected to external line loads at the crest and a stabilizing berm at the toe.

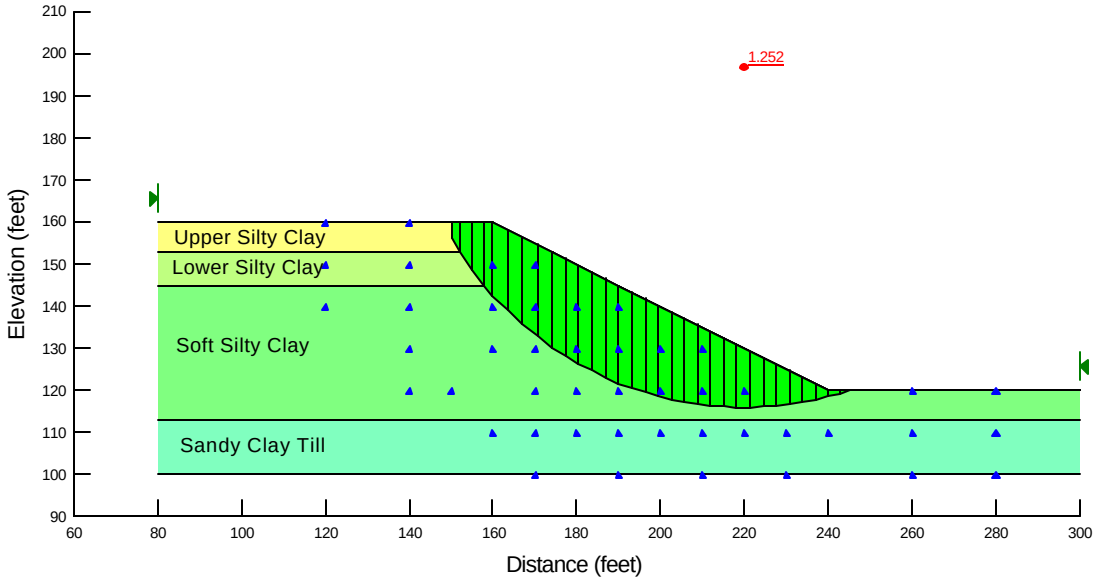
Figure 1.3 Example of a External Loads and Reinforcements



Complex Pore-Water Pressure Condition

Pore-water pressure conditions can be specified in a variety of ways. It may be as simple as a piezometric line or as complex as importing pore-water pressure conditions from a finite element analysis. Another procedure allows you to define the pore-water pressure conditions at a series of points as shown in Figure 1.4. The pore-water pressure at the base of each slice is determined from the data points by spline interpolation, (Kriging), techniques.

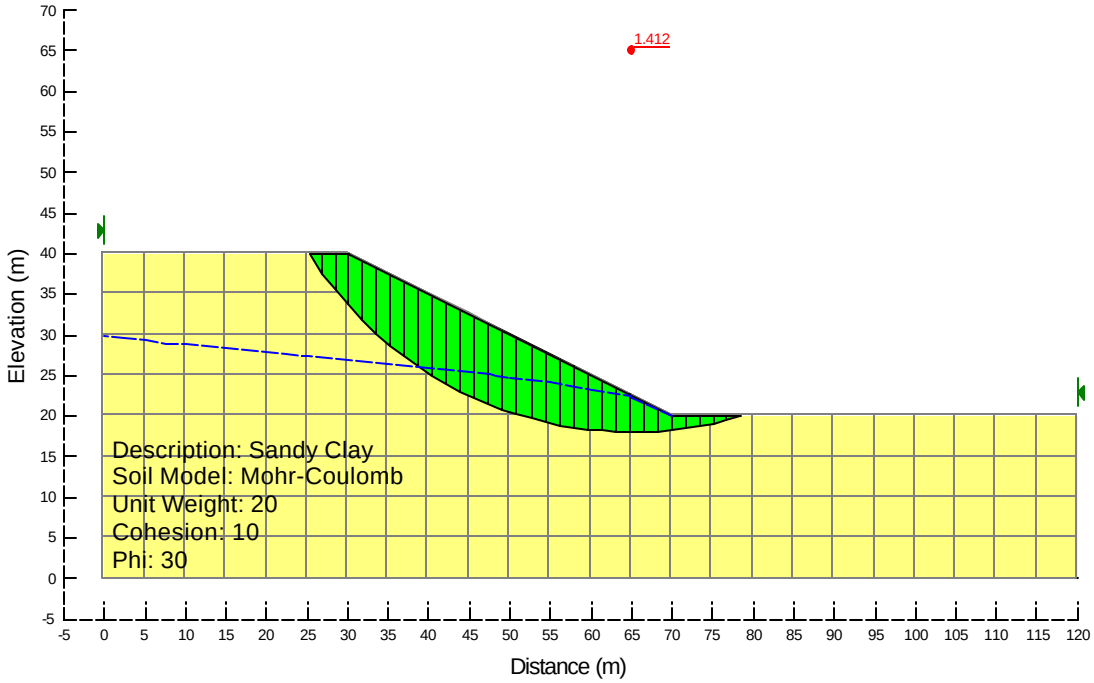
Figure 1.4 Example of a Slope with Complex Pore-Water Pressure Condition



Stability Analysis Using Finite Element Stress

The primary unknown in a slope stability analysis is the normal stress at the base of each slice. An iterative procedure is required to find the normal stress such that the factor of safety is the same for each slice and each slice is in force equilibrium. This iterative procedure can be avoided by importing the slope stresses into SLOPE/W from a SIGMA/W finite element stress analysis. SIGMA/W is another GEO-SLOPE product for stress and deformation analysis. The advantage of using finite element computed stresses is that it allows the calculation of the factor of safety for each slice, as well as the overall factor of safety for the slope. Figure 1.5 shows a stability analysis performed using SIGMA/W computed stresses.

Figure 1.5 Example of a Stability Analysis Using Finite Element Stress



Probabilistic Stability Analysis

Some degree of uncertainty is always associated with the input parameters of a slope stability analysis. To accommodate parameter uncertainty in the analysis, SLOPE/W has the ability to perform a Monte Carlo probabilistic analysis. In these cases, each input parameter is specified together with a standard deviation value to define a probability distribution for each input parameter. The standard deviation given for a particular parameter quantifies the degree of uncertainty associated with the parameter. Doing a probabilistic analysis makes it possible to compute a factor of safety probability distribution, a reliability index, and the probability of failure. The probability of failure is defined as the probability that the factor of safety is less than 1.0. The factor of safety is shown as a probability density function in Figure 1.6 and as a probability distribution function in Figure 1.7.

Figure 1.6 Results of Probabilistic Analysis Displayed as a Probability Density Function

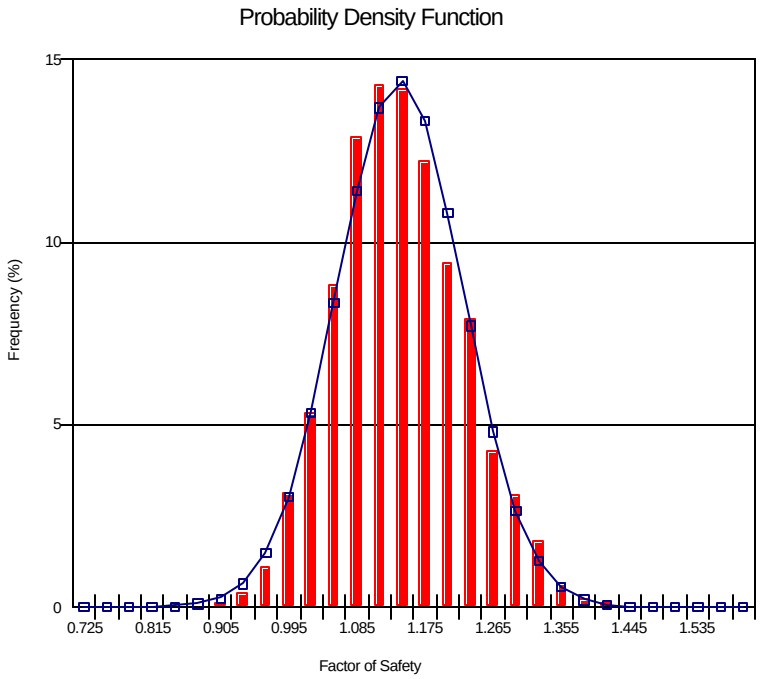
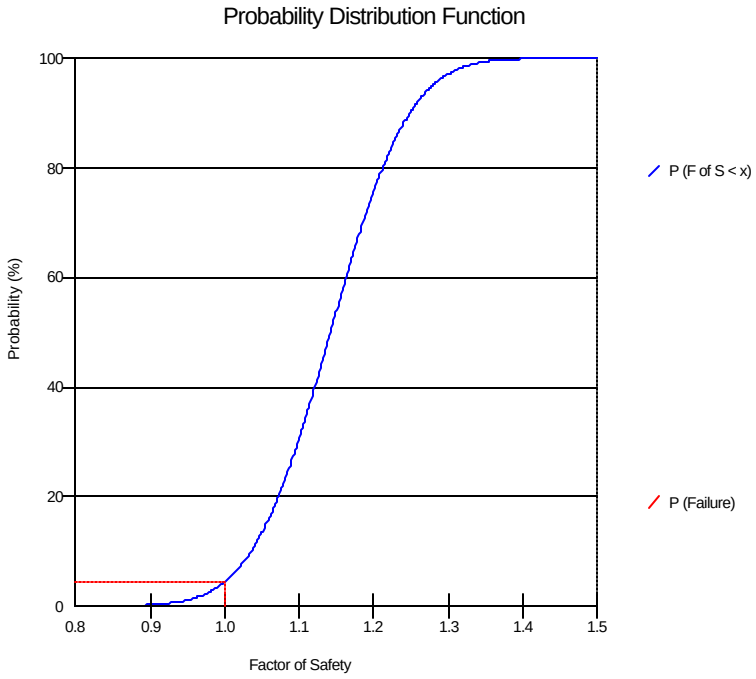


Figure 1.7 Results of Probabilistic Analysis Displayed as a Probability Distribution Function



Features and Capabilities

User Interface

Problem Definition

CAD is an acronym for Computer Aided Drafting. GEO-SLOPE has implemented CAD-like functionality in SLOPE/W using the Microsoft Windows graphical user interface. This means that defining your problem on the computer is just like drawing it on paper; the screen becomes your "page" and the mouse becomes your "pen." Once your page size and engineering scale have been specified, the cursor position is displayed on the screen in the actual engineering coordinates. As you move the mouse, the cursor position is updated. You can then "draw" your problem on the screen by moving and clicking the mouse.

The following are some of the model definition interface features:

- Display axes, snap to a grid, and zoom.

  To facilitate drawing, *x* and *y* axes may be placed on the drawing for reference. Using the mouse, axes may be selected then moved, resized or deleted. For placing the mouse on precise coordinates, a background grid may be specified. Using a “snap” option, the mouse coordinates will be set to exact grid coordinates when the mouse cursor nears a grid point. To view a smaller portion of the drawing, it is possible to zoom in by using the mouse to drag a rectangle around the area of interest. Zooming out to a larger scale is also possible.

- Sketch graphics, text and import picture.

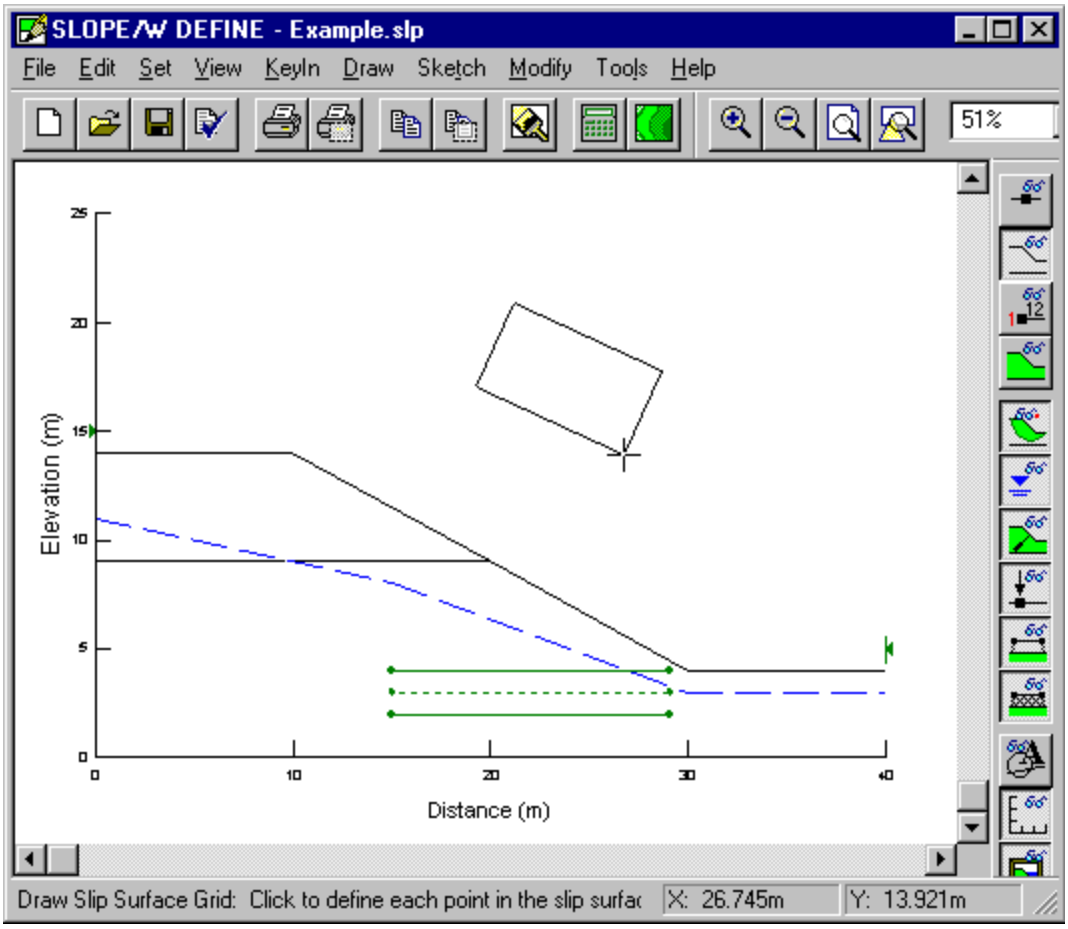
  Graphics and text features are provided to aid in defining models and to enhance the output of results. Graphics such as lines, circles and arcs, are useful for sketching the problem domain before defining a finite element mesh. Text is useful for annotating the drawing to show information such as material names and properties among other things. A dynamic text feature automatically updates project information text, soil property text and probabilistic analysis results text, whenever this information changes. This ensures that the text shown on the drawing always matches the model data.

  The import picture feature is useful for displaying graphics from other applications in your drawing. For example, a cross-section drawing could be imported from a drafting application for use as a background graphic while defining the problem domain. This feature can also be used to display things like photographs or a company logo on the drawing. Pictures are imported as a Windows metafile, (WMF), an enhanced metafile, (EMF), or a Windows bitmap file, (BMP).

  Using the mouse, individual or groups of graphics and text objects may be selected, then moved, resized or deleted.

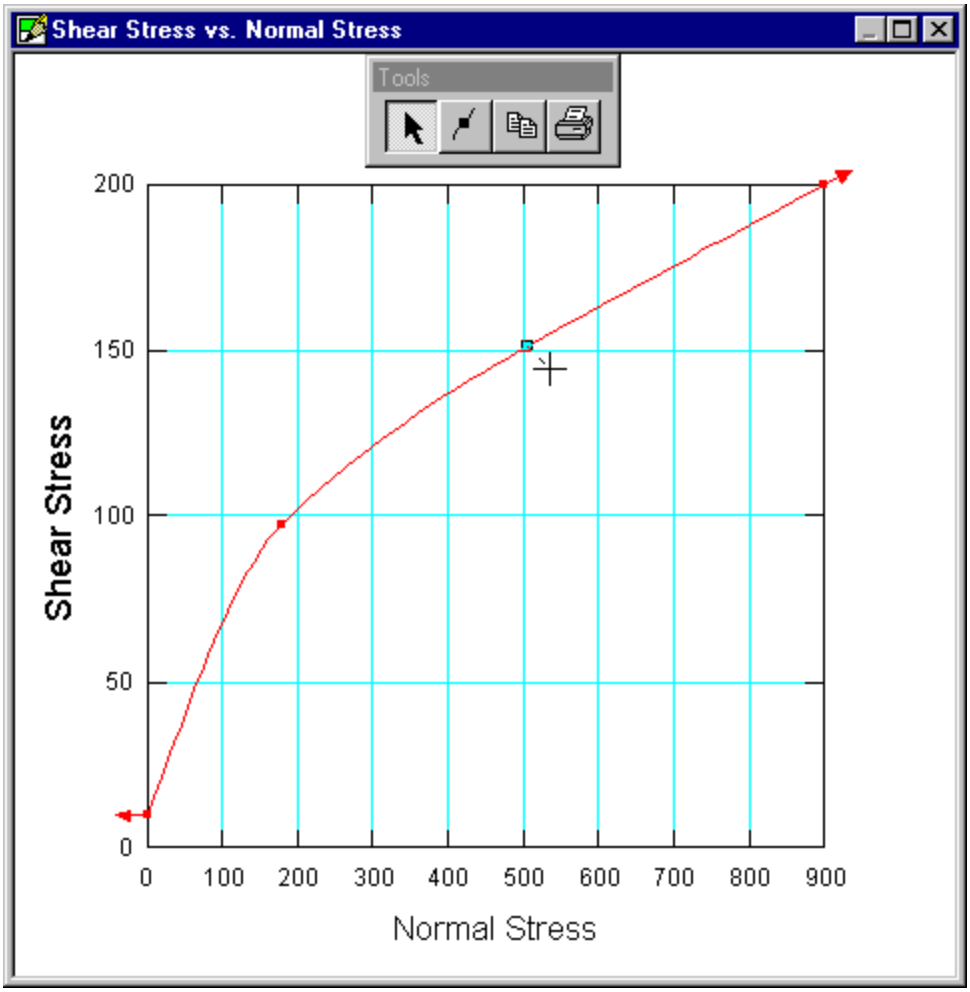
- Graphical problem definition and editing.

  Soil layer geometry, slip surfaces, pore-water pressure conditions, application of extemal loads and reinforcement, and tension zone location, can all be specified using the mouse. Individual or groups of these objects may be moved or deleted using the mouse to select and drag the objects. The figure below shows a grid of circular slip surface center points being defined using the mouse.


SLOPE/W DEFINE - Example.slp
File Edit Set View KeyIn Draw Sketch Modify Tools Help
51%
Elevation (m)
Distance (m)
Draw Slip Surface Grid: Click to define each point in the slip surfac
X: 26.745m
Y: 13.921m

- Graphical and keyboard editing of functions.

  SLOPE/W makes extensive use of functions. For example, the shear strength of a soil can be defined as a function of normal stress, or as a function of slice base inclination angle. All these functions can be edited graphically using the mouse and exact numerical values can be input using the keyboard. The figure below shows a point on a strength function being moved using the mouse.



Computing Results

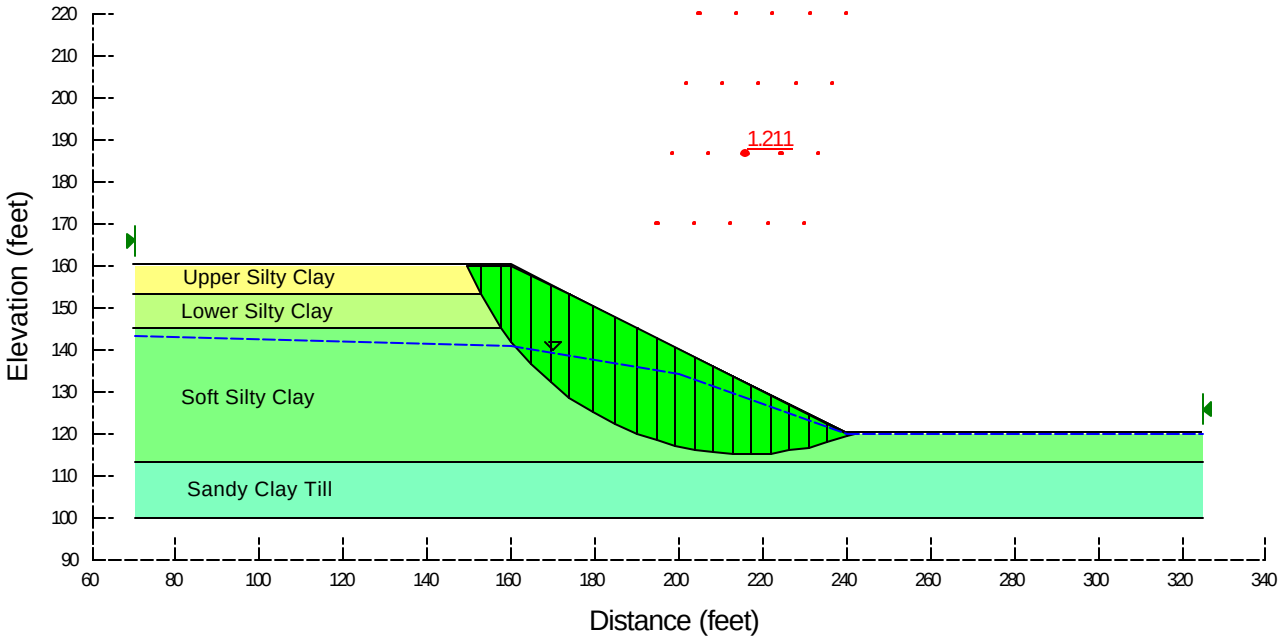
SLOPE/W computes the factor of safety for all specified trial slip surfaces. For probabilistic analyses, the Monte Carlo technique is used to compute the distribution of minimum factor of safety.

Viewing Results

After your problem has been defined and the solution computed, you can interactively view the results graphically. The following features allow you to quickly isolate the information you need from the computed data:

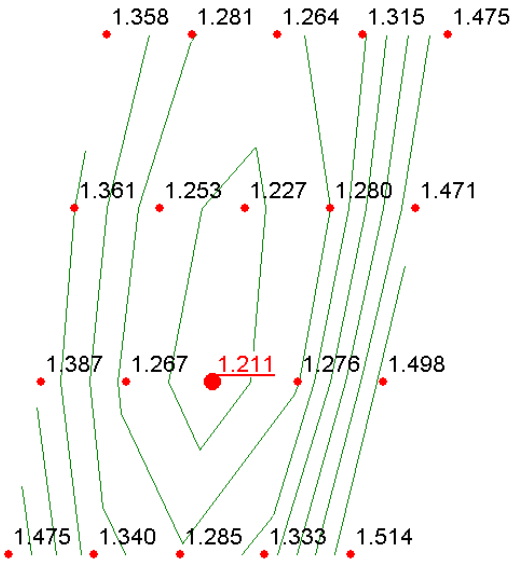
- View factor of safety and the associated critical slip surface.

  You can view the minimum factor of safety and the associated critical slip surface together. Factors of safety and the other non-critical slip surfaces can also be viewed. The figure below shows the critical slip surface and its factor of safety for the specified slope.



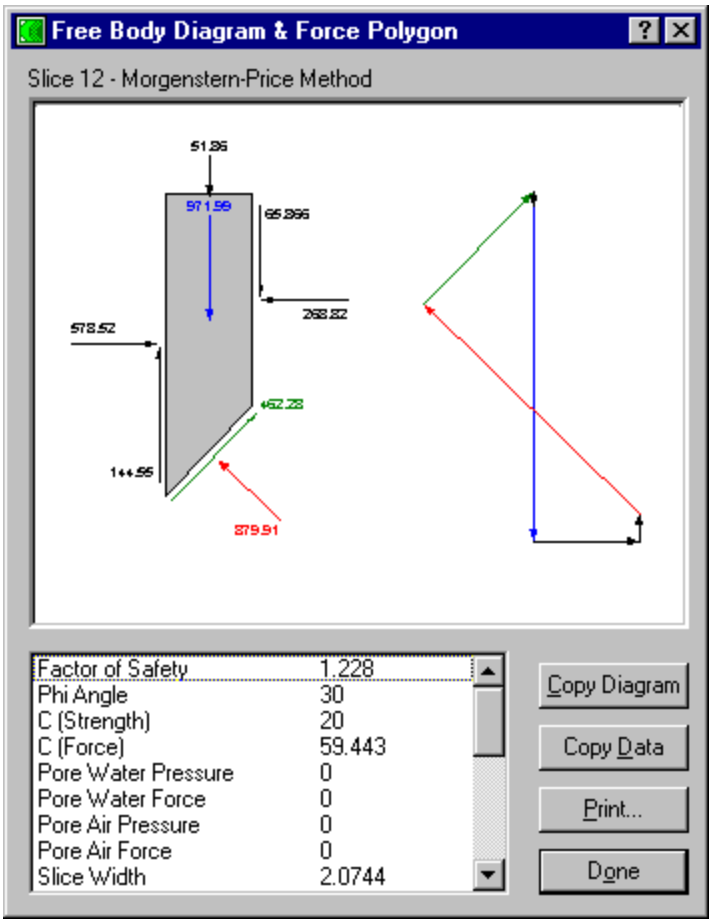
- Contour factor of safety values.

  To specify circular slip surfaces, a search grid of circular slip surface centers is defined. For each grid point, a series of trial radii are used to compute the lowest factor of safety value for the grid point. When the computations are complete, each grid point has a computed factor of safety value associated with it. The grid point with the lowest factor of safety represents the center of the critical circular slip surface. It is possible to contour the factor of safety values at the grid points, as shown in the figure below.



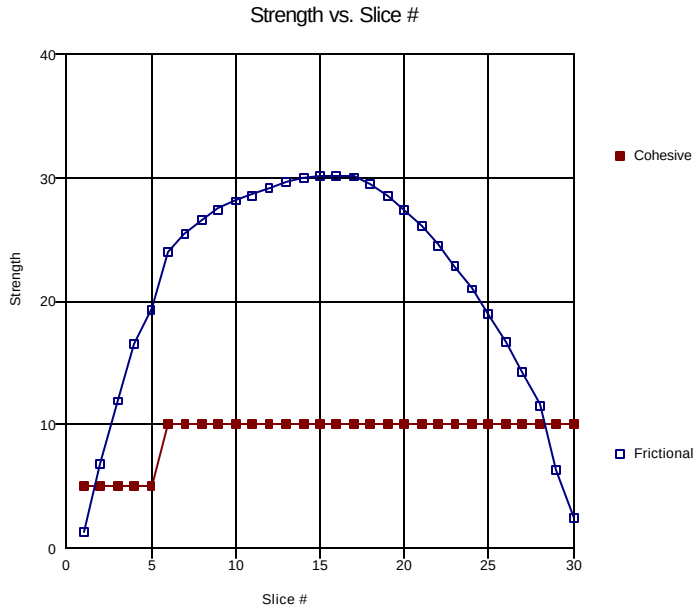
- View slice forces.

or each slice of the critical slip surface, the computed forces can be displayed as a free body diagram and force polygon along with the numerical force values. The figure below shows the forces on a single slice.



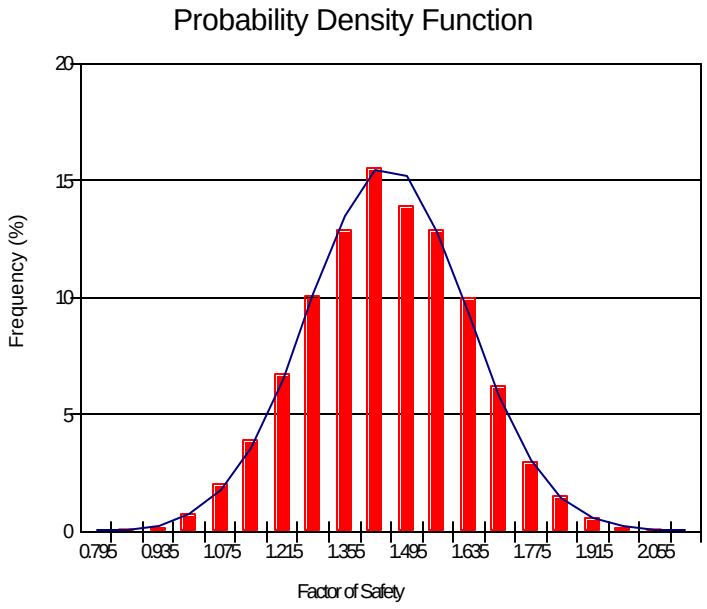
- Graph computed values along slip surface.

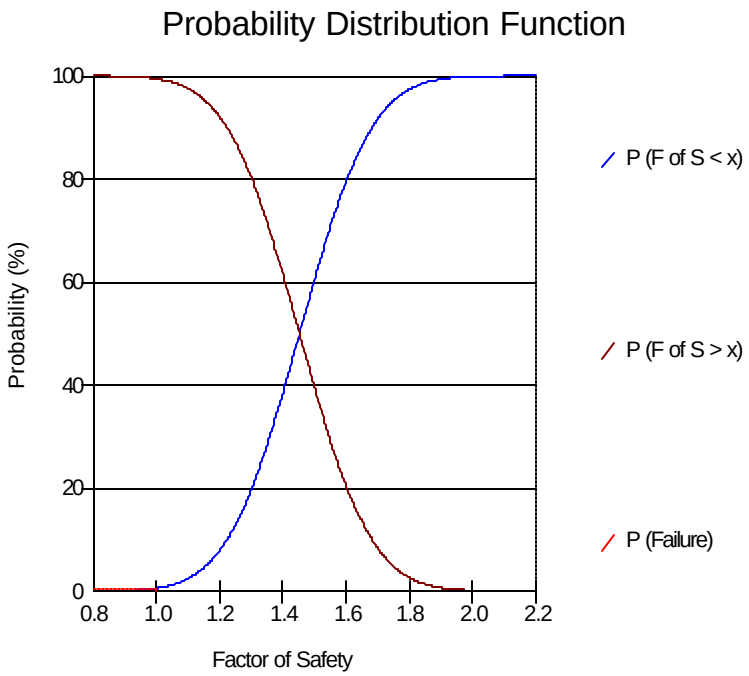
  ll computed values along the slip surface from crest to toe can be plotted on an *x-y* graph. This is useful for checking that the computed results are reasonable. The following figure shows a plot of cohesive and frictional strength at the base of each slip surface slice.



- Graph probability distributions.

  esults of probabilistic analyses can be displayed as a histogram or a cumulative frequency plot as shown in the figures below.





- Export computed data and graphics.

  To prepare reports, slide presentations, or add further enhancements to the graphics, SLOPE/W has support for exporting data and graphics to other applications. Computed data can be exported to other applications, such as spreadsheets, using ASCII text or using the Windows clipboard. The Windows enhanced metafile format, (EMF), and the Windows metafile format, (WMF), are supported for exporting graphics. For converting a WMF or EMF file to other file formats such as DXF, third party file format conversion programs can be used.

Other Interface Features

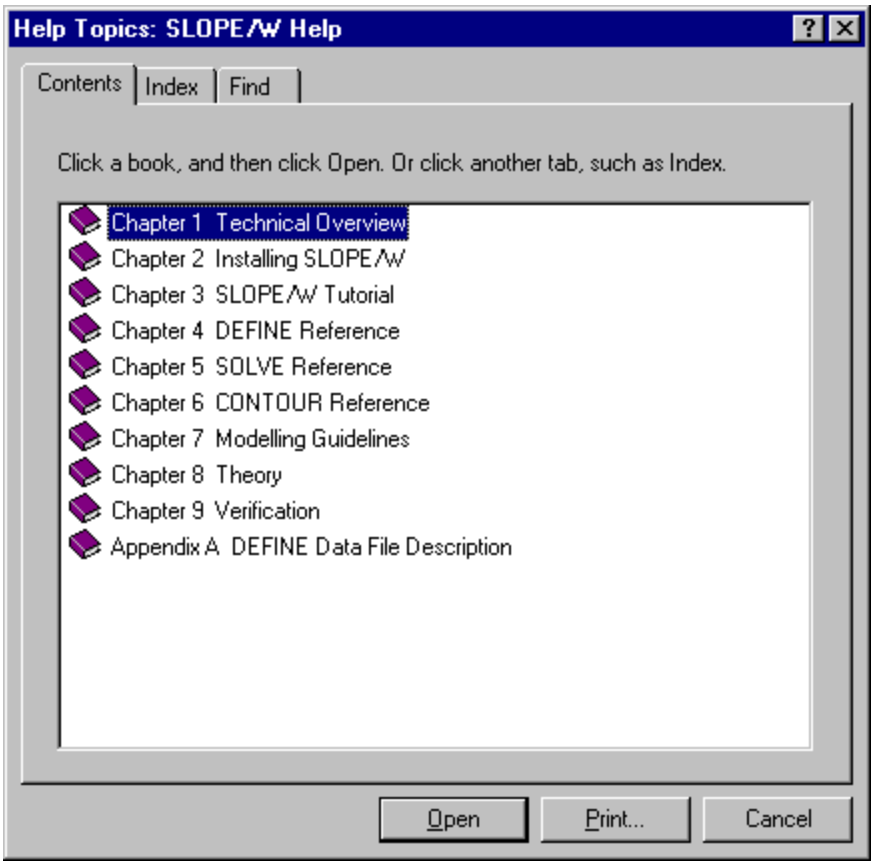
In addition to the features listed for model definition, computation, and viewing of results, the user interface has many other features commonly found in Windows applications. These are:

- Context sensitive help.

  All user interface items such as menu items, toolbars and dialog boxes provide context sensitive help. For example when a dialog box is displayed, hitting the F1 key will display a help topic related to that dialog box.

- On-line documentation.

  The on-line documentation contains the entire manual in the form of a Windows help file. This provides fast access to technical information and facilitates searching the manual for specific information. Each chapter of the on-line documentation is also available on the distribution CD-ROM as Microsoft Word documents that you can view or print.

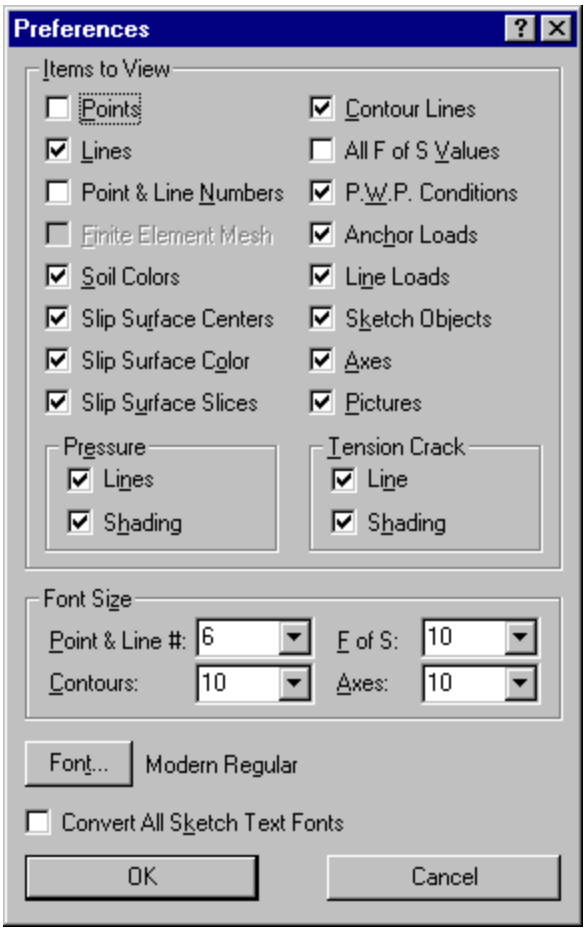


- Toolbar shortcuts for all menu commands.

  Toolbars contain buttons that provide a shortcut for all menu commands. The dockability of the toolbars mean that they can be repositioned and hidden according to your preferences.

- Extensive control on view preferences.

  View preference control allows you to display different types of objects on the drawing at the same time. Examples of these objects are shown in the figure below. All object types are displayed by default; however, you can turn off object types that you do not wish to view. This command also can be used to change the default font used for the problem, as well as the font size used for text, labels and axes.



- Designed for Windows 95 and Windows NT

  Because SLOPE/W was designed for Windows 95 and Windows NT, it has the common look and feel of other applications built for these operating systems. For example, SLOPE/W supports file names longer than eight characters, a most-recently-used file list for fast opening of recently used files, and common dialog boxes for common operations such as opening, saving and printing files.

Slope Stability Analysis

Analysis Methods

The comprehensive formulation of SLOPE/W allows stability analysis using the following methods:, Ordinary (or Fellenius) method, Bishop Simplified method, Janbu Simplified method, Spencer method, Morgenstern-Price method, Corps of Engineers method, Lowe-Karafiath method, generalized limit equilibrium (GLE) method, finite element stress method. Furthermore, a variety of interslice side force functions can be used with the more mathematically rigorous Morgenstern-Price and GLE methods.

The finite element stress method uses the stress computed from SIGMA/W, (a finite element software product available from GEO-SLOPE), to determine a stability factor. All the other methods use the limit equilibrium theory to determine the factor of safety.

The large selection of available analysis methods in SLOPE/W is provided so that you can decide which method suits the problem.

Probabilistic Analysis

SLOPE/W can perform probabilistic slope stability analyses to account for variability and uncertainty associated with the analysis input parameters. A probabilistic analysis allows you to statistically quantify the probability of failure of a slope using the Monte Carlo method. The results from all Monte Carlo trials can then be used to compute the probability of failure and generate the factor of safety probability density and distribution functions. Variability can be considered for material parameters such as unit weight, cohesion and friction angles, pore-water pressure conditions, applied line loads, and seismic coefficients.

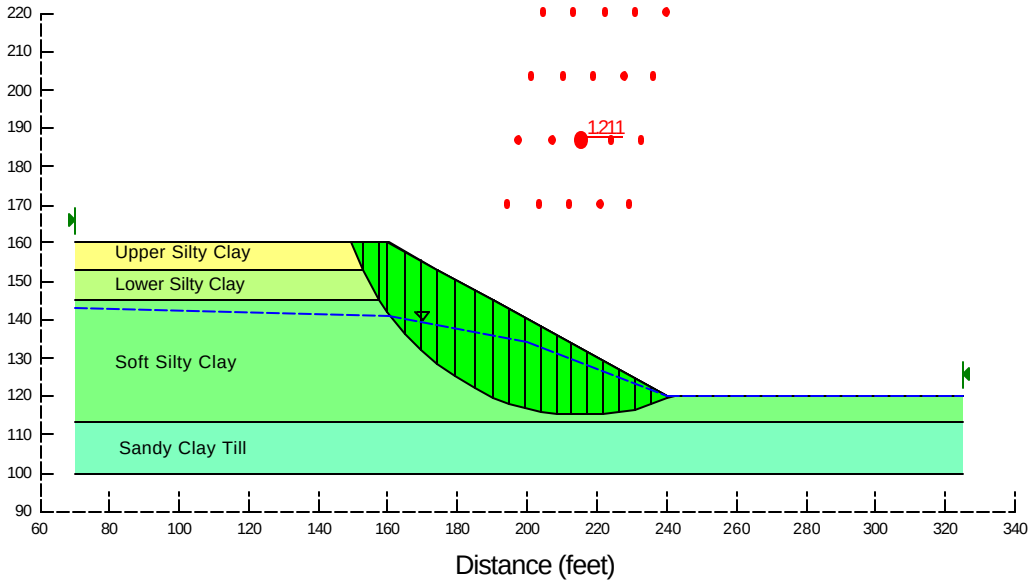
Geometry and Stratigraphy

SLOPE/W can be used to model a wide variety of slope geometry and stratigraphy such as multiple soil types, partial submergence in water, variable thickness and discontinuous soil strata, impenetrable soil layers, and dry or water-filled tension cracks. Tension cracks can be modelled with a specified tension crack line or a maximum slip surface inclination angle.

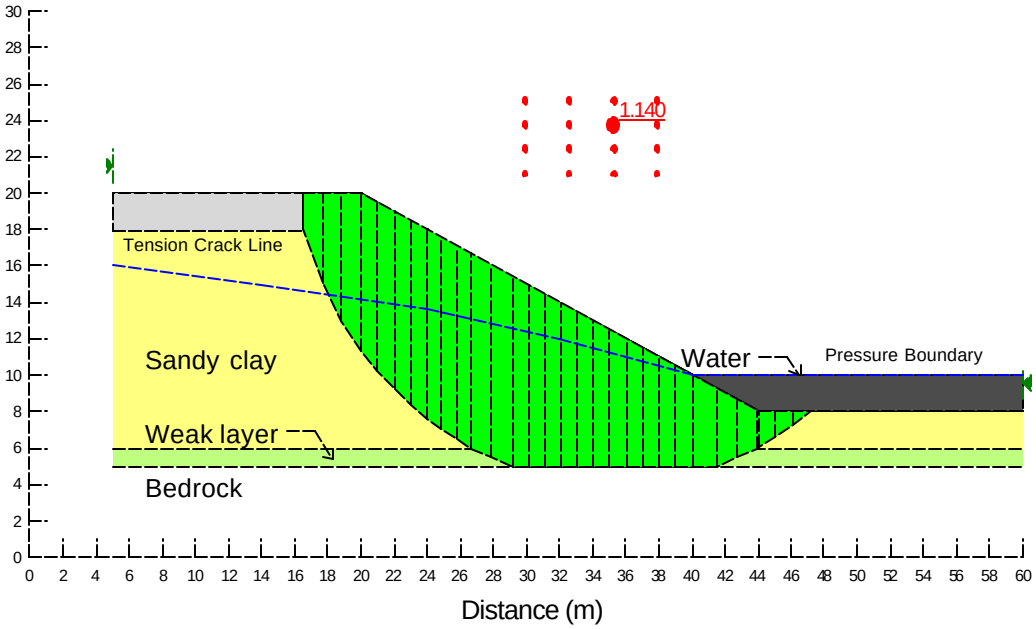
Slip Surfaces

SLOPE/W uses a grid of rotation centers and a range of radii to model circular and composite slip surfaces. SLOPE/W also provides block specified slip surface, and fully specified slip surface methods for modelling non-circular slip surfaces. The following figures illustrate the types of slip surfaces that can be modelled using SLOPE/W.

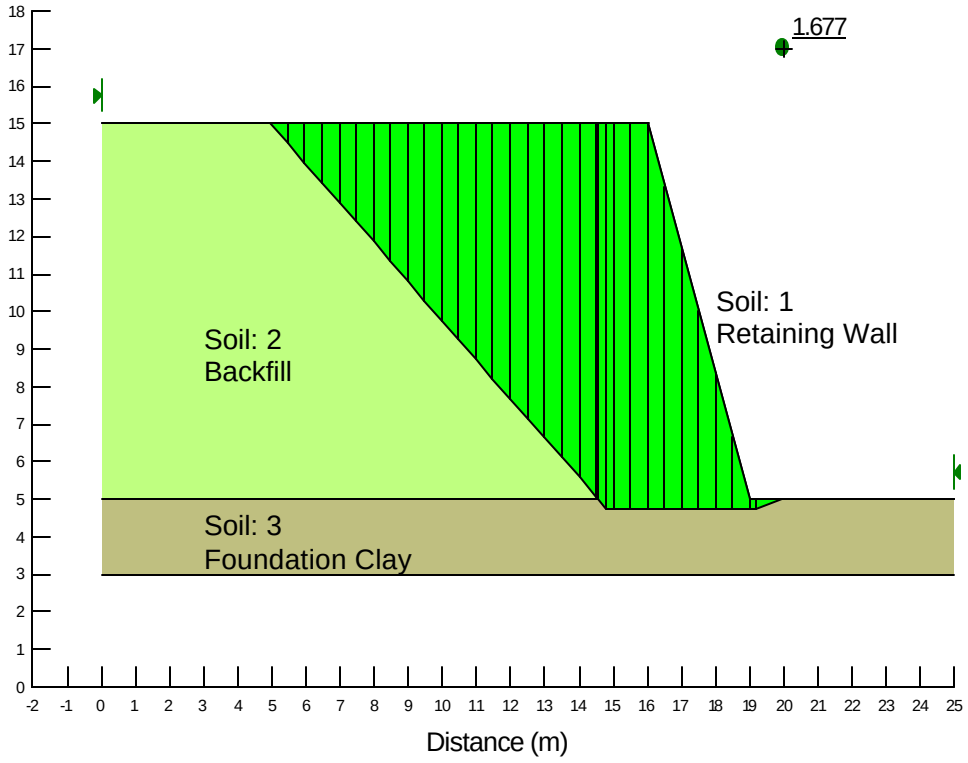
- Circular slip surface.



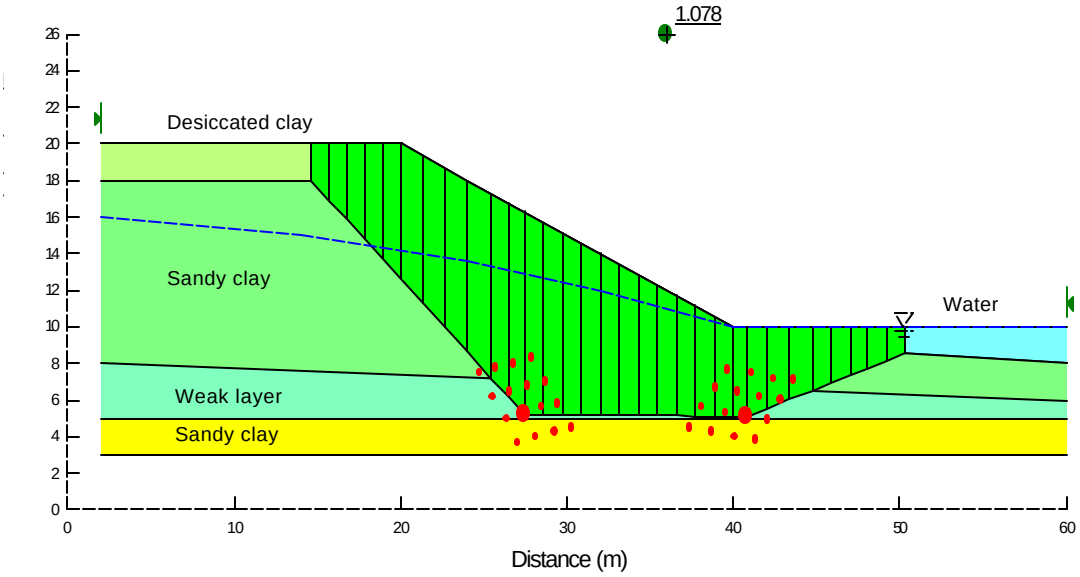
- Composite slip surface.



- Fully specified slip surface.



- Block specified slip surface.



Pore-Water Pressures

SLOPE/W provides many options to specify pore-water pressure conditions. Pore-water pressures can be defined as follows:

- Pore-water pressure coefficients.

  The classic pore water pressure coefficient, r_u, which relates the overburden stress to pore-water pressure, can be specified for each soil type.

- Piezometric surfaces.

  The easiest way to specify pore-water pressure conditions is to define a piezometric surface through the problem domain. For less common, non-hydrostatic situations, such as an artesian sand layer overlain by an clay aquitard, it is possible to define a separate piezometric surface for each soil layer.

- Pore-water pressure parameters at specific locations.

  If pore-water pressure parameters such as pressure, head, or r_u coefficients are known at specific locations within the soil, they can be specified in the model. This feature is useful for incorporating known field data into the analysis or for specifying complex pore-water pressure conditions. Spline interpolation of the specified data is used to calculate the pore-water pressure throughout the problem domain.

- Finite element computed pore-water pressures.

  SLOPE/W has the ability to import pore-water pressure data computed by SEEP/W or SIGMA/W, two of GEO-SLOPE's finite element programs. This capability is especially useful for performing slope stability analyses where the groundwater flow conditions are transient and/or significantly affected by the stress state within the soil.

- Pore-water pressure contours.

  If contours of pore-water pressure distribution are known, perhaps from field observations or some other type of seepage modelling, they can be used to specify the pore-water pressure conditions for a slope stability analysis.

Soil Properties

SLOPE/W provides the following material models to define the soil shear strength.

- Total and/or effective stress parameters.

  The Mohr-Coulomb parameters for cohesion and friction angle are the most common way to model soil shear strength. These parameters can be specified for either total or effective stress conditions in SLOPE/W.

- Undrained shear strength.

  Undrained soils exhibit no frictional shear resistance. The undrained soil model in SLOPE/W accommodates this by setting the friction angle, f, to zero.

- Zero shear strength materials.

  For materials which contribute only their weigh but add no shear strength to the system, SLOPE/W provides a zero shear strength material. Examples of zero shear strength materials include ponded water at the toe of a slope and surcharge fills. These materials have zero cohesion, (c=0), and zero friction angle, $(f = 0)$.

- Impenetrable materials.

  For the purposes of slope stability analyses, material through which a slip surface cannot penetrate are referred to as impenetrable materials. Where a slip surface encounters an impenetrable material such as bedrock, the slip surface continues along the upper boundary of the impenetrable material.

- Bilinear failure envelope.

  A bilinear Mohr-Coulomb failure envelope is useful for modelling materials that exhibit a change in frictional angle at a particular normal stress.

- Increasing cohesive shear strength with depth.

  In normally consolidated or slightly overconsolidated soils, cohesion increases with depth. SLOPE/W can accommodate these situations in two ways. The first way is by allowing the cohesive shear strength to vary with the depth below the top layer of the soil. This is useful for the analysis of natural slopes. The second way is by allowing the cohesive shear strength to vary as a function of elevation, independent of the depth from the top layer. This is useful for the analysis of excavated slopes.

- Anisotropic shear strength.

  Bedding planes in soil strata result in anisotropic shear strength values for cohesion and friction angle. SLOPE/W has a variety of ways to model anisotropic shear strength parameters, reflecting the variety of engineering practices used throughout the world.

- Custom shear strength envelope.

  In cases where a linear or bilinear Mohr-Coulomb failure envelope is insufficient for modelling soil shear strength, SLOPE/W provides the capability to specify a general curved relationship between shear strength and normal stress. This is the most general way to specify shear strength.

- Shear strength based on normal stress but with an undrained strength maximum.

  With this model, the shear strength is based on cohesion and friction angle up to a maximum undrained shear strength. Both cohesion and friction can vary with either depth below ground surface or with elevation above a datum.

- Shear strength based on the overburden effective stress.

  Soil shear strength in this model is directly related to the overburden effective stress by a specified factor, therefore increasing linearly with depth below the ground surface.

Applied Loads

Several kinds of external applied loads can be modelled using SLOPE/W. These include surcharge fill and structural loads, toe berm loads, line loads, anchor loads, soil nail loads, geo-fabric loads, and seismic loads.

Implementation

- 32-bit processing.

  32-bit processing allows full utilization of the CPU in current personal computers. Compared to 16-bit processing, 32-bit processing can result in a computational speed increase by a factor of two to three times, depending on problem size, number of iterations and number of time steps.

- No specific limits on problem size.

  SLOPE/W has been implemented using dynamic memory allocation, so there is no specific limits on problem size. Therefore the maximum size of the problem is a function of the amount of available computer memory.

Using SLOPE/W

SLOPE/W includes three executable programs; DEFINE, for defining the model, SOLVE for computing the results, and CONTOUR for viewing the results. This section provides an overview of how to use these programs to perform slope stability analyses.

Defining Problems

The DEFINE program enables problems to be defined by drawing the problem on the screen, in much the same way that drawings are created using Computer Aided Drafting, (CAD), software packages.

To define a problem, you begin by setting up the drawing space. This is done by setting a page size, a scale and the origin of the coordinate system on the page. Default values are available for all of these settings. To orient yourself while drawing, coordinate axes and a grid of coordinate points may be displayed.

Once the drawing space is specified, you can begin to sketch your problem on the page using lines, circles and arcs. You can additionally import a background picture to perform the same function. Having a sketch or picture of the problem domain helps to define the stratigraphy of the slope problem.

After defining the drawing space and displaying the problem domain, you then must specify material properties, define the slope geometry with points and lines, define the trial slip surfaces, specify the pore-water pressure conditions and apply the loading conditions. Most of these tasks can be performed with the mouse using commands on the Draw menu. Figure 1.8 shows the command available on the Draw menu. Material property values are keyed into dialog boxes using commands available under the KeyIn menu.

Figure 1.8 also shows a few of the user interface features designed to make the software easier to use. Toolbars contain button shortcuts for commonly used menu commands. DEFINE has five toolbars, each for different groups of commands. A status bar, located at the bottom of the window shows the mouse position in engineering coordinates.

Figure 1.9 shows the end result of defining the slope stability model. The slope geometry has been defined, material properties have been assigned, trial slip surfaces have been defined, and pore-water pressure conditions applied. Saving the problem creates a DEFINE data file to be read in by the SOLVE program. After this is done, the problem is ready to be solved.

Figure 1.8 Problem Domain Displayed in SEEP/W DEFINE Window

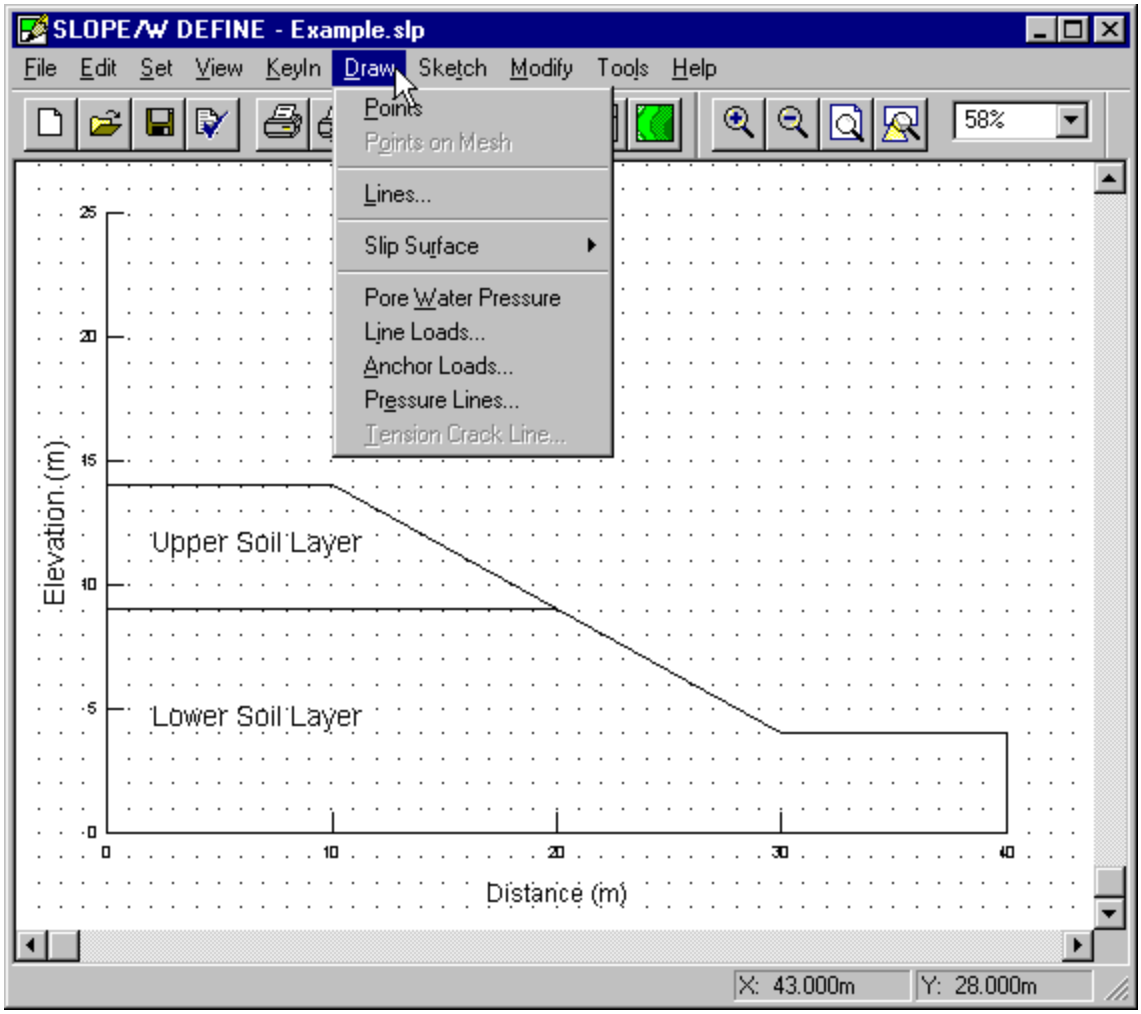
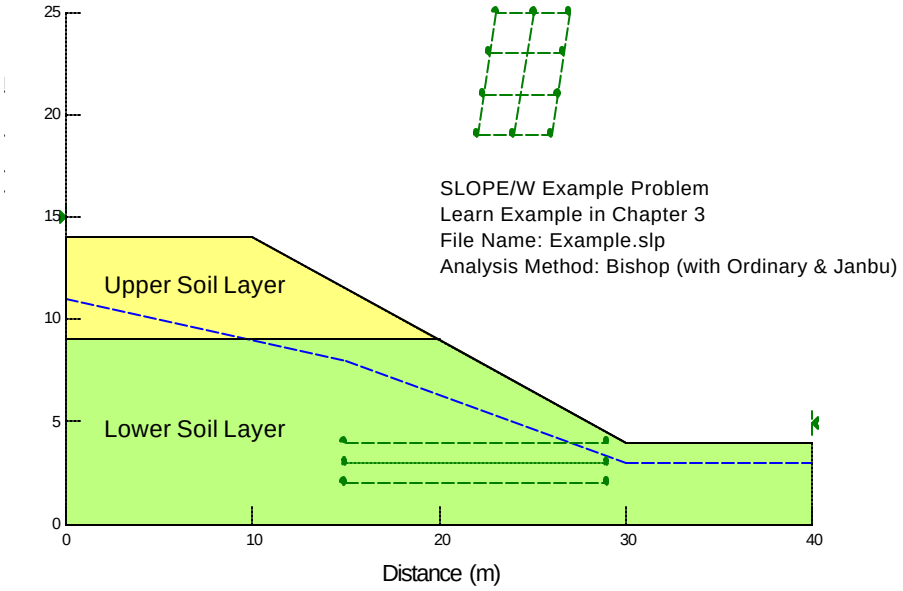


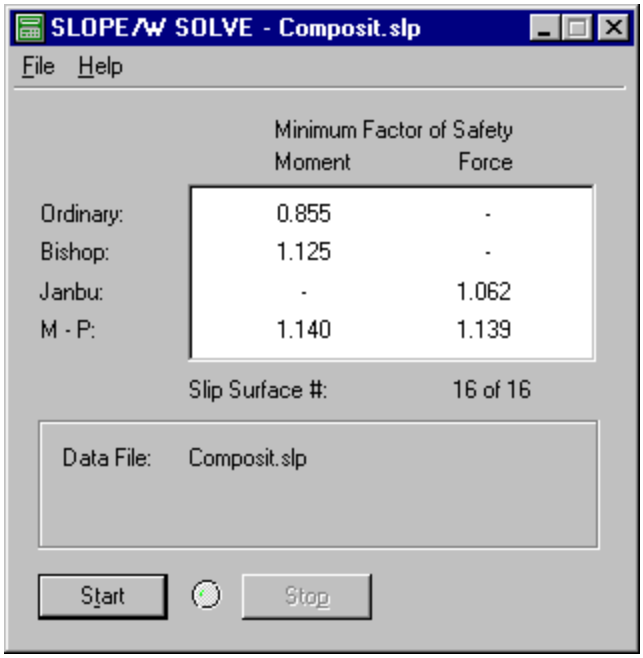
Figure 1.9 Fully Defined Slope Stability Problem



Solving Problems

Once a data file is created with DEFINE, the problem is solved using the SOLVE program. Figure 1.10 shows the main window of the SOLVE program with a DEFINE data file opened. Pressing the Start button begins the computations. Information is displayed in the large list box area during the computations. The computations can be stopped at any time.

Figure 1.10 SOLVE Main Window



Contouring and Graphing Results

CONTOUR graphically displays all the trial slip surfaces and the factors of safety computed by SOLVE. The results may be presented as factor of safety contours, force diagrams and force polygons for individual slices, graphs of computed parameters along the slip surface, and factor of safety probability distributions.

The CONTOUR program has the same CAD-like features as DEFINE and operates in a similar fashion. Data review is accomplished using commands on the View and Draw menus, shown in Figures 1.11 and 1.12, respectively. The View menu contains commands oriented towards viewing the factor of safety computed using various methods, viewing numerical information for points and soil properties, and viewing forces on individual slices. The Draw menu contains commands oriented towards presenting the results graphically. The computed factor of safety of any trial slip surface can be displayed. The computed factor of safety can be contoured and labelled. Computed quantities of each slice along the critical slip surface can be graphed as a function of the distance along the slip surface or as a function of the slice number.

In addition to data visualization, the drawing can be enhanced and labelled with graphics and text. Objects can be selected with the mouse and then moved, resized or deleted.

Figure 1.11 View Menu in CONTOUR

Figure 1.12 Draw Menu in CONTOUR

Formulation

SLOPE/W is formulated in terms of two factor of safety equations. These equations are used to compute the factor of safety based on slice moment and force equilibrium. Depending on the interslice force function adopted, the factor of safety for all the methods can be determined from these two equations.

One key difference between the various methods is the assumption regarding interslice normal and shear forces. The relationship between these interslice forces is represented by the parameter l. For example, a l value of 0 means there is no shear force between the slices. A l value that is nonzero means there is shear between the slices.

Figure 1.13 Plot of Factor of Safety vs. Lambda (l)

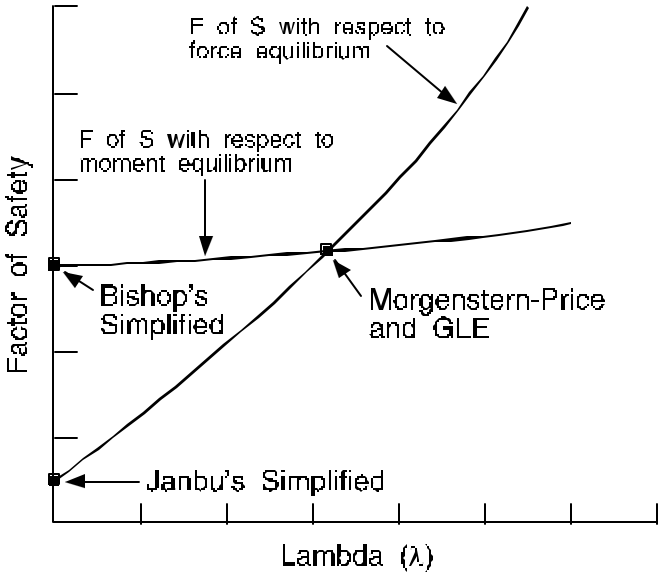

F of S with respect to force equilibrium
F of S with respect to moment equilibrium
Factor of Safety
Bishop's Simplified
Morgenstern-Price and GLE
Janbu's Simplified
Lambda (λ)

Figure 1.**Error! Bookmark not defined.** presents a plot of factor of safety versus l. Two curves are shown in the figure. One represents the factor of safety with respect to moment equilibrium, and the other one represents the factor of safety with respect to force equilibrium. Bishop's Simplified method uses normal forces but not shear forces between the slices $(l = 0)$ and satisfies only moment equilibrium. Consequently, the Bishop factor of safety is on the left vertical axis of the plot. Janbu's Simplified method also uses normal forces but no shear forces between the slices and satisfies only force equilibrium. The Janbu factor of safety is therefore also on the left vertical axis. The Morgenstern-Price and GLE methods use both normal and shear forces between the slices and satisfy both force and moment equilibrium; the resulting factor of safety is equal to the value at the intersection of the two factor of safety curves. The illustration in Figure 1.**Error! Bookmark not defined.** shows how the general formulation of SLOPE/W makes it possible to readily compute the factor of safety for a variety of methods.

In addition to the limit equilibrium methods of analysis, SLOPE/W also provides an alternative method of analysis using the stress state obtained from SIGMA/W, a GEO-SLOPE program for finite element stress and deformation analysis. The stability factor of a slope using the finite element stress method is defined as the ratio of the summation of the available resisting shear force along a slip surface to the summation of the mobilized shear force along a slip surface. The mobilized shear force along a slip surface is calculated based on the computed stress state from SIGMA/W. The normal stress at the base of each slice is also obtained from SIGMA/W and is then used to calculate the available resisting shear force along the slip surface.

SLOPE/W can perform probabilistic slope stability analyses for any of the limit equilibrium and finite element stress methods using the Monte Carlo technique. The critical slip surface is initially determined based on the mean value of

the input parameters. Probabilistic analysis is then performed on the critical slip surface, taking into consideration the variability of the input parameters. The variability of the input parameters is assumed to be normally distributed with user-specified mean values and standard deviations.

During each Monte Carlo trial, the input parameters are updated based on a normalized random number. The factors of safety are then computed based on these updated input parameters. By assuming that the factors of safety are also normally distributed, SLOPE/W determines the mean and the standard deviations of the factors of safety. The probability distribution function is then obtained from the normal curve.

Product Integration

GEO-SLOPE provides the following suite of geotechnical and geo-environmental engineering software products:

- SLOPE/W for slope stability
- SEEP/W for seepage
- CTRAN/W for contaminant transport
- SIGMA/W for stress and deformation
- TEMP/W for geothermal analysis

SLOPE/W is integrated with SEEP/W and SIGMA/W. The integration of this geotechnical software allows you to use results from one product as input for another product. Examples of the integration between products are listed below.

- The SEEP/W computed head distribution can be used in SLOPE/W slope stability analyses, which is particularly powerful in the case of transient processes. Using the SEEP/W results for each time increment in a SLOPE/W stability analysis makes it possible to determine the factor of safety as a function of time.

  Consider, for example, the changing pore-water pressure conditions in an embankment as the excess pressures dissipate after reservoir drawdown. SEEP/W can compute the pore-water pressure at various times after reservoir drawdown. The conditions at each time can be used in a slope stability analysis, making it possible to establish the margin of stability as a function of time after the start of the drawdown.

- Pore-water pressures that arise due to external loading can be computed by SIGMA/W as part of a stress analysis. SLOPE/W can use the SIGMA/W-computed stress-induced excess pore-water pressures in a stability analysis. This makes it possible, for example, to compute the end-of-construction stability conditions in terms of effective stresses.

- SIGMA/W-computed finite-element stresses can be used in SLOPE/W to compute stability factors. This new and innovative method makes it possible to assess the overall stability of a slope as well as the local stability factor of each slice.

Product Support

You may contact GEO-SLOPE in Calgary to obtain additional information about the software. GEO-SLOPE's product support includes assistance with resolving problems related to the installation and operation of the software. Note that the product support does *not* include assistance with modelling and engineering problems.

GEO-SLOPE updates the software periodically. For information about the latest versions and available updates, visit our World Wide Web site.

http://www.geo-slope.com

If you have questions or require additional information about the software, please contact GEO-SLOPE using any of the following methods:

E-Mail:

support@geo-slope.com

Phone:

403-269-2002

Fax:

403-266-4851

Mail or Courier:

GEO-SLOPE International Ltd.

Suite 1400, Ford Tower

633 - 6th Avenue S.W.

Calgary, Alberta, Canada

T2P 2Y5

GEO-SLOPE's normal business hours are Monday to Friday, 8 a.m. to 5 p.m., Mountain time.

Chapter 2 Installing SLOPE/W

Basic Windows Skills **3**
- Windows Fundamentals 3
- Locating Files and Directories 3
- Viewing Data Files 4
- System Messages 5

Basic SLOPE/W Skills **6**
- Starting and Quitting SLOPE/W 6
- Dialog Boxes in SLOPE/W 6
- Using Online Help 8

Installing the Software **9**
- Using the CD-ROM 9
- Installing GEO-SLOPE Evaluation Software 9
- Viewing GEO-SLOPE Manuals 9
- Installing SLOPE/W 10
  - Running the CD-ROM Setup 10
  - The Security Key 11
  - The Sentinel System Drivers 13
  - Files Installed by Setup 14
  - Starting SLOPE/W 15

Installing Additional Software for Network Versions **16**
- The Rainbow NetSentinel Software 16
- Network Version Requirements 17
- Installing the Rainbow Network Software 17
- Security Server Reference 18
  - Choosing the Security Computer 18
  - Running the Security Server on a NetWare File Server 21
  - Running the NetSentinel Service Security Server under Windows NT 23
  - Running the Win32 Windows Security Server 27
  - Running the Security Server on a DOS Computer 30
  - Running the Security Server on an OS/2 Computer 33
- Security Monitor Reference 36
  - The Security Monitor Programs 36
  - Running WINMON, the Windows-Based Security Monitor 36
  - Running DOSMON, the DOS-Based Security Monitor 38
  - Running OS2MON, the OS2-Based Security Monitor 39
- NetSentinel Configuration Reference 41
  - Banyan Vines 41
  - IBM LAN Server/Requester 2.x and 3.x 42
  - LANTastic 43
  - Microsoft LAN Server/Requester 2.0 and 2.1 44
  - Novell NetWare 3.x and 4.x 45
  - Windows for Workgroups 3.11 (NetBEUI) 45
  - Windows NT / Windows NT with Novell NetWare 46
  - Windows 95 / Windows 95 with Novell NetWare 47
  - Using TCP/IP with Windows 95 and NT 48

Basic Windows Skills

Windows Fundamentals

To install and use SLOPE/W, you must first install Microsoft Windows 95 or Windows NT Version 3.51 (or greater) and be familiar with its operation. The Microsoft Windows documentation will help you in learning how to use Windows. Since the SLOPE/W documentation does not fully cover the Windows operating instructions, you may need to use both the Windows and the SLOPE/W documentation while you are getting started.

The fundamentals of using Windows are explained in detail in the Microsoft Windows documentation, which gives hints on choosing commands in Windows and using dialog boxes.

All SLOPE/W commands are accessed from the menu bar or from toolbars. To choose a SLOPE/W menu command with the mouse, click on the menu name, and then click on the name of the command in the drop-down menu. A short description of the command is displayed in the SLOPE/W status bar as you move the mouse over the menu item. To choose a menu command from the keyboard, press ALT to select the menu bar, and use the arrow keys to move to the command; press ENTER to choose the command. Alternatively, press ALT, and then press the underlined letter of the menu name. When the drop-down menu is displayed, press the letter of the command.

To choose a SLOPE/W toolbar command, click on the desired toolbar button. If you hold the cursor above the toolbar button for a few seconds, the command name is displayed in a small “tool-tip” window.

Commands are named according to the menu titles. For example, the File Open command is so named because it is accessed by selecting the File menu from the menu bar and then choosing Open from the File menu.

Some drop-down menu commands contain a triangle on the right side. This means that there is a cascading menu with additional commands. An example of this type of command is the KeyIn Functions command found in SLOPE/W DEFINE.

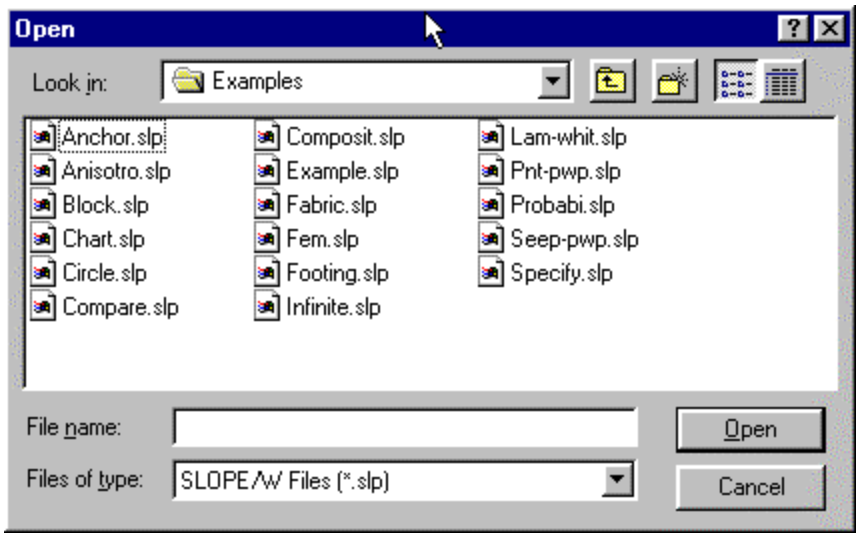
Many SLOPE/W commands use dialog boxes to obtain additional information from you. Dialog boxes contain various options, each asking for a different piece of information. To move to a dialog option using the mouse, click on the option. To move to the next option in the sequence using the keyboard, press TAB. Press SHIFT+TAB to move to the previous option.

Command buttons are options in dialog boxes that initiate an immediate action. For example, a button labelled **OK** accepts the information supplied by the dialog box, while a button labelled **Cancel** cancels the command. To choose a button with the mouse, click on the button. To choose a button from the keyboard, select the button by moving to it with the TAB key. A dark border appears around the currently selected, or default, button. Press ENTER to choose this button. The Cancel button can be chosen from the keyboard by pressing ESC.

Locating Files and Directories

Knowing how to locate files and directories is essential to learning how to use SLOPE/W.

The following is a typical File Open dialog box:



➢ **To open a file:**

- Type a name in the File Name edit box and then select Open. The file name may include a directory and a path.

  -- or --

- click on a file name in the list box and then select Open.

  -- or --

- double-click on a file name in the list box.

➢ **To change the current directory or drive:**

- Select the directory name from the Look In drop-down list box.
- Select a new drive from the Look In drop-down list box or select the Network Neighborhood option to connect to a network drive, if available.

➢ **To view a specific group of file names within the current directory:**

- Select the file group to display in the Files of Type drop-down list box.

  -- or --

- type the file name, including any wild card characters (*****, **?**), in the File Name edit box and click the Open button. For example, typing ***.SLP** will list all the files in the current directory with a file extension of **SLP**.

Viewing Data Files

SLOPE/W saves all data files in ASCII text format, allowing you to view the files with any text editor. To view SLOPE/W data files while running Windows, you will need to be familiar with the Windows accessories WordPad and Notepad. For more information about these applications, see the Windows documentation. For information about the DEFINE data file format, see the DEFINE Data File Description appendix.

System Messages

Windows displays system messages in a message box whenever it cannot execute a command or a warning needs to be issued about a command. For example, Windows 95 displays a message indicating that the drive is not accessible or that the device is not ready.

Whenever you are asked to make a choice before proceeding, the message box will contain two buttons, such as a Retry button and a Cancel button. You can proceed by clicking the Retry button or you can negate the command by clicking the Cancel button.

When SLOPE/W encounters a command that it cannot process, Windows terminates SLOPE/W and displays an error message. This may happen if you try to read an incompatible data file. Also, it may happen if there is an error in SLOPE/W, such as dividing a number by zero. Please contact GEO-SLOPE if you cannot determine the reason for an error message.

Basic SLOPE/W Skills

Starting and Quitting SLOPE/W

➢ **To start SLOPE/W in Windows 95 or Windows NT 4.0:**

1. Click the Start button or press Ctrl-Esc to open the Start menu.
2. Move the mouse pointer to the Programs folder and select the SLOPE/W folder from the menu.
3. Click on the appropriate SLOPE/W program icon to start DEFINE, SOLVE, or CONTOUR.

➢ **To start SLOPE/W in Windows NT 3.51:**

1. Double-click the SLOPE/W program group icon in the Program Manager window.
2. Double-click on the appropriate SLOPE/W program icon to start DEFINE, SOLVE, or CONTOUR.

➢ **To Quit SLOPE/W:**

- Choose File Exit from the SLOPE/W menu.

  -- or --

- With the SLOPE/W window selected, press Ctrl+F4.

  -- or --

- Right-click the title bar of the SLOPE/W window and select Close.

  -- or --

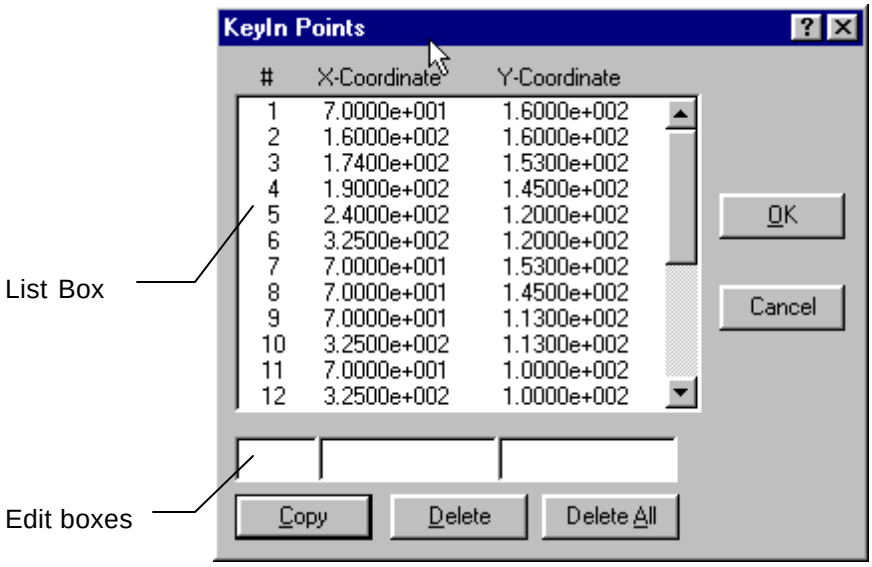
- Double-click the control menu box in the upper-left corner of the SLOPE/W window.

For more details on starting and quitting applications from Windows, refer to your Windows documentation.

Dialog Boxes in SLOPE/W

SLOPE/W uses many dialog boxes for entering and editing lists of numeric data. An example of this type of dialog box is illustrated in Figure 2.1, which contains a dialog box used for entering and modifying the list of points.

Figure 2.1 A Dialog Box for Entering and Modifying Points



New points are entered by typing the coordinates in the edit boxes and copying to the list box. Points are edited by copying data from the list box to the edit boxes and making changes.

Copy Copies values from the edit boxes to the list box.

Delete Deletes the line of data that is highlighted in the list box.

Delete All Deletes all lines of data in the list box.

OK Saves the changes you have made to the values in the list box.

Cancel Ignores all entries and changes made to the dialog box and returns you to the previous state of the program.

➢ **To enter a new point in the list box:**

1. Type the point number and its coordinates in the edit boxes.
2. Select Copy.

   The new point is copied into the list box.

➢ **To change the data relating to an existing point:**

1. In the list box, click on the point to change.

   The line in the list box is highlighted, and the point number and its coordinates are automatically copied to the edit boxes.

2. Make the necessary changes in the edit boxes.
3. Select Copy.

   The point is copied into the list box, replacing the point that has a point number matching the value contained in the # edit box.

➢ **To delete a point from the list box:**

1. In the list box, click on the point to delete.
2. Select Delete.

   The point is removed from the list box.

Dialog boxes of this type may have other controls, such as a View button. See the appropriate online help section for details on each specific dialog box.

Using Online Help

The SLOPE/W Online Help system provides you with a powerful means of accessing the SLOPE/W documentation. It gives you several different ways to answer your SLOPE/W questions:

- Browse the SLOPE/W Contents to see a hierarchical display of all SLOPE/W Help Topics
- View an alphabetical index of SLOPE/W Help Topics
- Search for all SLOPE/W Help Topics that contain a specific word or phrase
- Display the SLOPE/W Help Topic for the SLOPE/W dialog box or command you are currently using.

➢ **You can access SLOPE/W Help in the following different ways:**

- Choose Help Topics from the SLOPE/W Help menu.

  A Help Topics dialog box is displayed containing Contents, Index and Find tabs. Select these tabs to browse the SLOPE/W Help Contents, view the SLOPE/W Help Index, or search for a SLOPE/W help topic using a word or phrase.

- Press the F1 key.

  A Help Topics dialog box is displayed containing Contents, Index and Find tabs.

- Move the mouse over a SLOPE/W menu item (such that the menu command is highlighted) and press the F1 key.

  SLOPE/W displays the help topic corresponding to the selected menu command.

- Press down on a SLOPE/W toolbar button and press the F1 key.

  SLOPE/W displays the help topic corresponding to the selected toolbar button.

- While you are in a SLOPE/W interactive mode, such as Draw Points, press the F1 key.

  SLOPE/W displays the help topic corresponding to the mode (e.g., the Draw Points help topic).

- While you are in a SLOPE/W dialog box, press the F1 key or press the question mark button in the top-right corner of the dialog box.

  SLOPE/W displays the help topic corresponding to the dialog box.

NOTE: You can get help on the Windows Help Topics dialog box by pressing the F1 key or the question-mark button in the top-right corner of the dialog box.

Installing the Software

Using the CD-ROM

SLOPE/W is distributed to you on the GEO-SLOPE Office CD-ROM. The CD-ROM contains a setup program that installs the SLOPE/W software on your computer.

- **To run the setup program:**

1. Insert the distribution CD-ROM into your CD-ROM drive.

2. If you are running Windows 95 or Windows NT 4.0 or higher, the Setup program is automatically loaded when the CD-ROM is inserted into the drive. Alternatively, from the Start Menu, you can select run and type **d:\setup** in the dialog box, where **d:** is your CD-ROM drive. Select OK to start the Setup program.

   GEO-SLOPE Office Version 4 software cannot be run under Windows 3.11. If you are running Windows 3.11 or earlier, you can use GEO-SLOPE Version 3 software.

3. Click on the View Installation Instructions option if you wish to display or print the setup instructions.

Installing GEO-SLOPE Evaluation Software

The GEO-SLOPE CD-ROM includes evaluation copies of each GEO-SLOPE Office software product: SLOPE/W, SIGMA/W, SEEP/W, CTRAN/W, and TEMP/W. The Evaluation Software is a feature-complete version of each product that you are free to copy and distribute; you can use it to examine, test and assess all features of the software. The only limitation of the Evaluation Software is that you cannot analyze your own specific problems. The Evaluation Software does not require a security key.

- **To install the GEO-SLOPE Office Evaluation Software:**

1. Run the main Setup program from the distribution CD-ROM.

2. Click on Install GEO-SLOPE Office Evaluation Software in the Setup window.

   The Evaluation Software Setup program begins execution.

3. Follow the instructions given by the Evaluation Software Setup program.

Viewing GEO-SLOPE Manuals

The SLOPE/W On-line Help system provides you with a powerful, interactive means of accessing the SLOPE/W documentation from within the software. While the Getting Started Guide contains Chapters 1 through 3 of the SLOPE/W Manual in printed form, you may wish to view and print other chapters of the manual.

The GEO-SLOPE Office CD-ROM contains the complete manuals for all GEO-SLOPE Office products in Microsoft Word format. All document (.doc) files can be viewed with Microsoft Word or the Microsoft Word Viewer included with the CD-ROM. If you have Microsoft Word on your system, you can use this software to view, read and print all or the document files on the CD-ROM; otherwise, you can install the Microsoft Word Viewer.

- **To install Microsoft Word Viewer:**

1. Run the main Setup program from the distribution CD-ROM.

2. Click on Install Microsoft Word Viewer in the Setup window.

   The Word Viewer Setup program begins execution.

3. Follow the instructions given by the Word Viewer Setup program.

➢ **To view and print the SLOPE/W manual in Word format:**

1. Start Microsoft Word Viewer or any word-processor capable of recognizing Word format files.

2. Choose File Open in the Word Viewer and load a chapter from the SLOPE/W manual.

   Each SLOPE/W chapter is located on the GEO-SLOPE CD-ROM under the directory \OFFICEV4\MANUALS\SLOPEW. The **Outline.doc** file provides an overview of the contents of each chapter.

3. Choose File Print in the Word Viewer to print the chapter.

4. Repeat Steps 2 to 3 for each chapter that you wish to print.

The above procedure can be used to print the manual for any other GEO-SLOPE Office software product.

Installing SLOPE/W

Running the CD-ROM Setup

To install SLOPE/W, you will need the GEO-SLOPE Office CD-ROM and a separate 3.5" SLOPE/W floppy diskette (the diskette is not required for installing the evaluation software). It is recommended that you make a backup copy of the SLOPE/W diskette before installation.

➢ **To install SLOPE/W:**

1. Run the main Setup program from the GEO-SLOPE Office CD-ROM.

2. Click on Install GEO-SLOPE Office Applications in the Setup window.

   The GEO-SLOPE Office Setup program begins execution.

3. Follow the instructions given by the Setup program.

   When you are prompted for the software products that you wish to install, select SLOPE/W. You will need the SLOPE/W 3.5" diskette to complete the installation.

Using the SLOPE/W 3.5" Diskette

After you select the GEO-SLOPE Office products that you wish to install (i.e., SLOPE/W), Setup will prompt you to insert the 3.5" distribution diskette for each product. If you do not have a local diskette drive but you are connected to a network, copy the contents of the SLOPE/W 3.5" diskette from a different computer to a directory on the network. When Setup prompts you to insert the SLOPE/W 3.5" diskette, specify the network directory as the path to the diskette.

➢ **To install SLOPE/W on a notebook computer with a removable CD-ROM drive:**

1. Insert the SLOPE/W 3.5" diskette in your floppy diskette drive.

2. Copy the contents of the SLOPE/W diskette into a new folder on your notebook hard drive (e.g., **c:\slopew\disk1**).

3. If your notebook computer only has one drive bay, turn off your computer and replace your floppy diskette drive with your CD-ROM drive.

4. Insert the GEO-SLOPE Office CD-ROM in your CD-ROM drive and run Setup to install SLOPE/W. When you are prompted for the SLOPE/W 3.5" diskette, type the SLOPE/W folder you created (in Step 2) on your notebook hard drive (e.g., **c:\slopew\disk1**).

5. Follow the remaining instructions in the Setup program to complete the installation.

Network Administrator Setup

If you are a Network Administrator installing SLOPE/W on more than one client computer, there are two ways you can install the software. The first method is to run Setup from the CD-ROM on each client computer, as described earlier in this chapter. The other method is to install SLOPE/W on your network and then have each user run Setup from the network; this method is called the Network Administrator Setup.

- **To run the Network Administrator Setup:**

1. Run Setup as described earlier and choose Network Administrator Setup when prompted.

2. Follow the remaining Setup instructions.

   All SLOPE/W software components are copied to the network directory that you specify; your Windows registry is not modified and no SLOPE/W folders are created on your desktop.

   A Client Setup program is also copied to the network directory; this is the program that is to be run by each user to install the software.

- **To run the Client Setup from a user workstation:**

1. Run Setup from the network directory specified in the Network Administrator Setup.

2. Follow the Setup instructions.

   You can choose to install SLOPE/W on your local workstation or to run it remotely from the network server. If you install SLOPE/W on your local workstation, all files will be copied to the directory you specify. If you choose to run SLOPE/W from the network, Setup will not copy any files from the network; it will only create a local SLOPE/W folder on the computer and modify the Windows registry. For a Client Setup, you do not need to use the SLOPE/W 3.5" diskette.

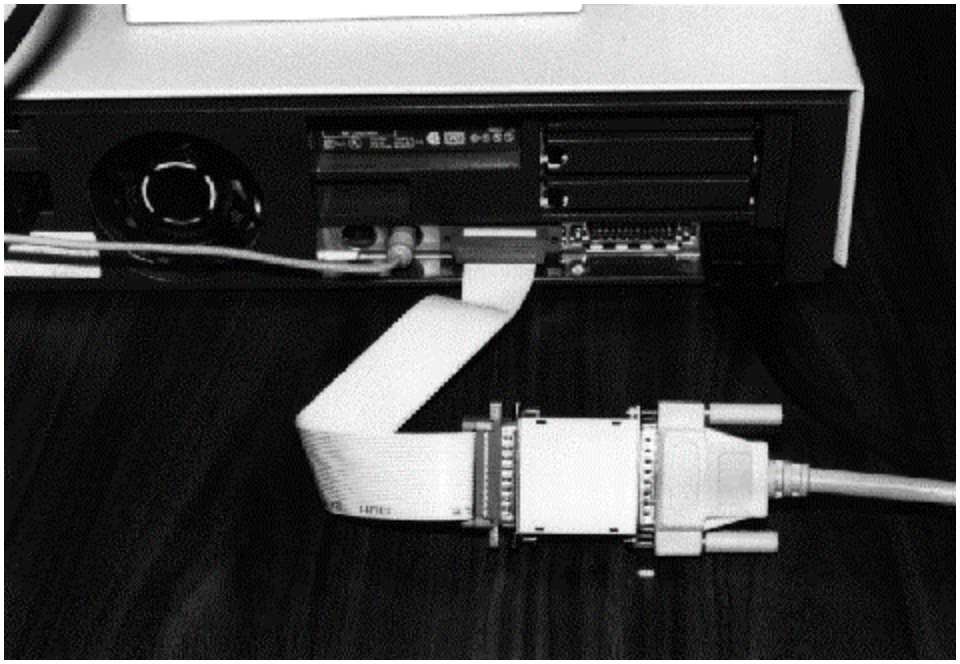
The Security Key

The execution of all GEO-SLOPE Office software is controlled by a small hardware device, called a Sentinel security key, as shown in Figure 2.2. In order to run a stand-alone version of SLOPE/W, you must have a Sentinel key attached to the computer's parallel (printer) communications port and the key must remain attached while using SLOPE/W.

A short ribbon cable is included in the software package to make it more convenient to attach any of the Sentinel keys, although it is not essential to use the cable. A parallel printer cable may be attached to the other end of the Sentinel key, allowing for normal printer operation even while the key is in place.

There are two types of Sentinel security keys: the SentinelPro for SLOPE/W stand-alone versions and the NetSentinel for SLOPE/W network versions. To enable a security key to work together with SLOPE/W, you will need to install additional software during setup.

Figure 2.2 The Sentinel Security Key Attached between a Ribbon Extension Cable and a Printer Cable



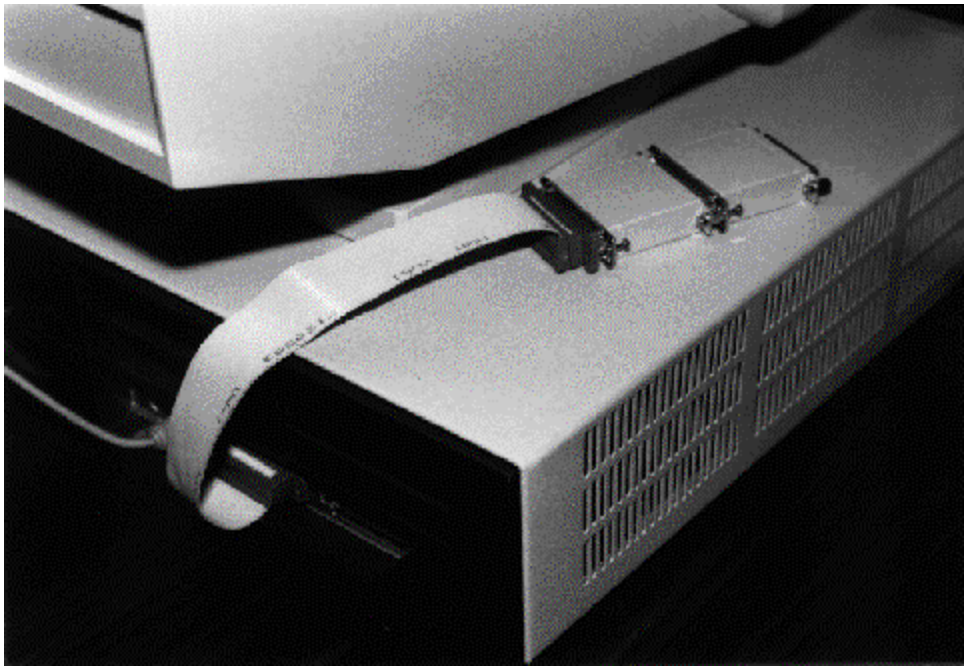
The SentinelPro Security Key

The SLOPE/W SentinelPro security key is supplied with the SLOPE/W stand-alone version. The SentinelPro key must be attached to your computer's parallel port, as described in the previous topic. If it is not attached, SLOPE/W will not be able to run.

There is a unique SentinelPro key for each GEO-SLOPE software product. If you have multiple GEO-SLOPE products, you may cascade the keys by connecting each key to the back of the other, as shown in Figure 2.3. This makes it possible to use multiple products without adding or removing the SentinelPro keys.

NOTE: If you setup a stand-alone version of SLOPE/W to run from the network server, you must still attach the SentinelPro to your own computer, *not* to the network server computer.

Figure 2.3 Two Cascaded SentinelPro Keys



The NetSentinel Security Key

The SLOPE/W NetSentinel network security key is supplied with the SLOPE/W Network Version. The NetSentinel security key monitors the number of users running SLOPE/W concurrently, as well as ensuring that properly licensed software is being used. The NetSentinel key must be attached to a designated computer somewhere on your network; this computer is referred to here as the *security computer*.

➢ **To run the SLOPE/W Network Version:**

1. Install SLOPE/W by choosing Install GEO-SLOPE Office Applications from the main Setup window.

2. Install additional network software on the security computer by choosing Install Additional Software for Network Versions from the main Setup window.

   To run the SLOPE/W Network Version, the NetSentinel key must be attached to the server computer, and the server computer must be running the installed *network security server* program.

NOTE: If you are the only user running SLOPE/W, you can alternatively connect the NetSentinel key to your computer's parallel port. This allows the NetSentinel key to be used like a stand-alone SentinelPro key; no one else on the network will be able to run SLOPE/W and you do not need to run the security server program on your computer.

The Sentinel System Drivers

A *Sentinel system driver* must be installed on the computer that has either the NetSentinel or SentinelPro security key attached to it. This driver allows Windows to communicate with the security key.

When you install SLOPE/W, you are asked if you wish to install the Sentinel system driver on your computer. If you are installing a stand-alone version of SLOPE/W, you will need to install this driver, since you must have the SentinelPro key attached to your computer to run SLOPE/W.

If you are installing the SLOPE/W Network Version, you will only need to install the Sentinel system driver if you are attaching the NetSentinel key to your computer's parallel port. For example, if you install the SLOPE/W Network Version on five different computers, you only need to install the Sentinel system driver on the one computer that has the NetSentinel key attached to it. You can also choose to attach the NetSentinel key to a computer that doesn't run SLOPE/W, such as your network file server. In this case, you can install the Sentinel system driver on your network server by choosing Install Additional Software for Network Versions from the main Setup window.

After you have installed the system driver, you have to reboot your computer before the driver can become operational. Setup will prompt you for rebooting at the conclusion of the installation process.

Different versions of the Sentinel system driver are included for following operating systems:

- ***Windows NT*** When you install the Sentinel driver under Windows NT, the NT version of the Sentinel system driver for Intel machines is installed. See the SENTINEL.HLP Windows Help file in the \SENTINEL\WIN_NT\I386 CD-ROM directory for a description of how the Sentinel system driver is installed.

  NOTE: If you are having difficulties installing the Sentinel driver under Windows NT, contact your system administrator to see if your account has sufficient rights to install drivers. You may need to log in as Administrator to install the Sentinel driver.

- ***Windows 95*** When you install the Sentinel driver under Windows 95, the Sentinel system driver for Windows 95 is installed. For more information, see the SENTW95.HLP Windows Help file in the \SENTINEL\WIN_95 CD-ROM directory for a description of how the Sentinel system driver is installed.

- ***OS/2*** Setup does not automatically install the OS/2 Sentinel system driver. If you need this driver in order to run the OS/2 NetSentinel security server, you will have to install it yourself from the GEO-SLOPE Office CD-ROM. See the README.TXT file in the \SENTINEL\OS2 CD-ROM directory for information on installing the OS/2 Sentinel system driver.

Files Installed by Setup

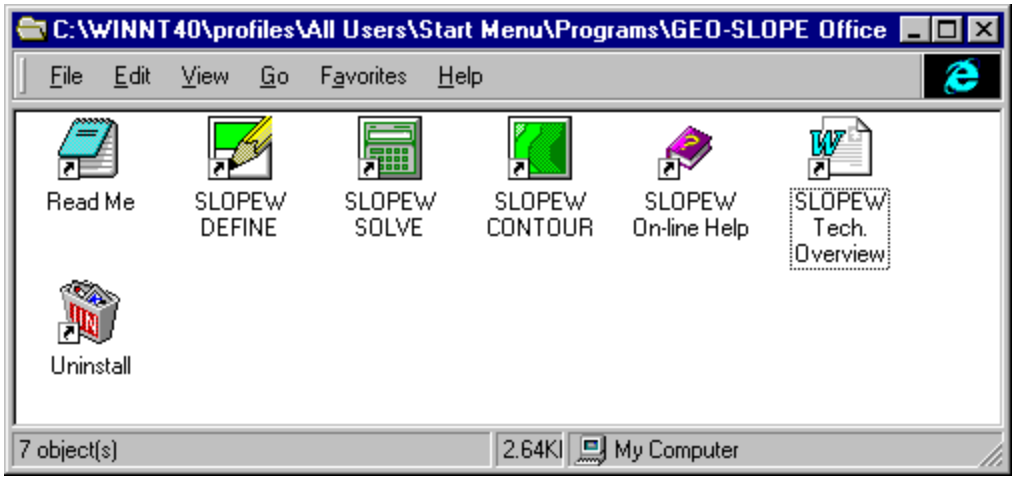
Table 2.1 describes the files installed by SLOPE/W Setup and specifies the default directories where they are installed.

Table 2.1 SLOPE/W Files and Installation Directories

SLOPE/W File	Description	Default Installation Directory
SLOPE1.EXE	SLOPE/W DEFINE	\GSI_VER4\SLOPEW
SLOPE2.EXE	SLOPE/W SOLVE	\GSI_VER4\SLOPEW
SLOPE3.EXE	SLOPE/W CONTOUR	\GSI_VER4\SLOPEW
SLPHLP.HLP	SLOPE/W Online Help	\GSI_VER4\SLOPEW
SLPHLP.CNT	SLOPE/W Online Help Contents	\GSI_VER4\SLOPEW
GSI0132.DLL	GEO-SLOPE DLL	\GSI_VER4\SLOPEW
NSLMS324.DLL	GEO-SLOPE DLL	\GSI_VER4\SLOPEW
GSW32.EXE	Graphics Server program	\GSI_VER4\SLOPEW
GSWAG32.DLL	Graphics Server DLL	\GSI_VER4\SLOPEW
GSWDLL32.DLL	Graphics Server DLL	\GSI_VER4\SLOPEW
Example Files	SLOPE/W example problems	\GSI_VER4\SLOPEW\EXAMPLES
Manual Files	User's Guide Word documents	\GSI_VER4\SLOPEW\MANUAL

Starting SLOPE/W

The following folder is created by the GEO-SLOPE Office Setup program when you choose the Typical Setup option:



To run SLOPE/W, double-click on one of the SLOPE/W modules in the folder. You can also run SLOPE/W from the Windows Start menu. See your Windows documentation for a description of changing the SLOPE/W folder properties.

Installing Additional Software for Network Versions

The Rainbow NetSentinel Software

The SLOPE/W Network Version makes it possible for you to use SLOPE/W on any computer in your network. It also allows a group of people to use the software simultaneously. For example, if you purchased a 5-user license of the SLOPE/W Network Version, up to 5 people on the network can use SLOPE/W concurrently.

The SLOPE/W NetSentinel network security key is supplied with the SLOPE/W Network Version. The NetSentinel security key monitors the number of users running SLOPE/W concurrently and ascertains that properly licensed software is being used. The NetSentinel key must be attached to a designated computer somewhere on your network; this computer is referred to here as the *security computer*.

The SLOPE/W Network Version requires additional software to manage the NetSentinel security key. This software is supplied to GEO-SLOPE by Rainbow Technologies, the makers of the NetSentinel key. The Rainbow NetSentinel software includes the following three items:

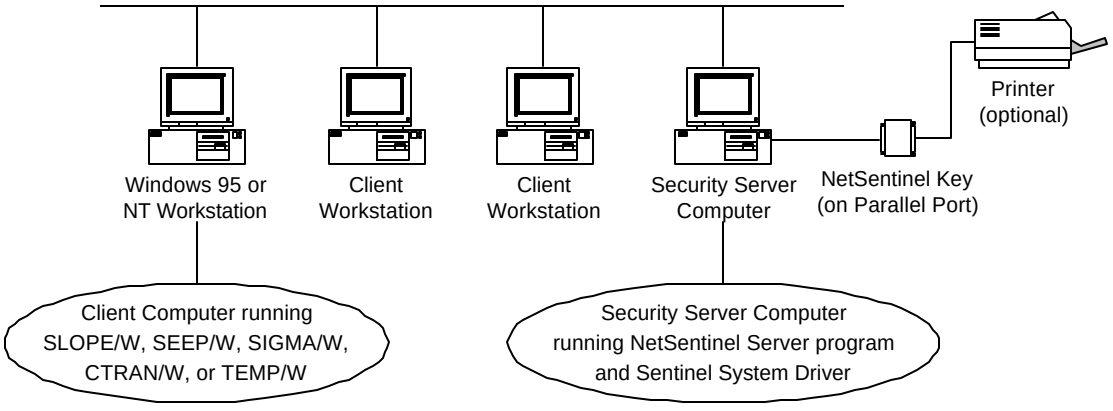
1. A *network security server* program must be run on the security computer; this program communicates with the NetSentinel key and keeps track of how many users are running SLOPE/W concurrently.

2. A Sentinel system driver is installed on the security computer; this driver allows the security server program to communicate with the NetSentinel key on the parallel port.

3. A *network security monitor* program can optionally be run from any computer on the network.

   This monitoring program displays information about the security server and security key. This information includes server transport protocols, the number of licenses in use, the number of users who were disconnected after timing out, and the license limit for each key. You do not need to install the security monitor to use the SLOPE/W Network Version; however, the security monitor is useful for administrating the network software.

The following diagram illustrates how the NetSentinel security key and software are connected to your network:



NetSentinel Security Key and Software Configuration

Network Version Requirements

The SLOPE/W Network Version can be run from the network file server or from each user's local hard disk, depending on your preference. The basic requirements for running the SLOPE/W Network Version are:

1. The computer network must support at least one of the following protocols: NetBIOS, IPX/SPX, Named Pipes, or TCP/IP.

   - If TCP/IP is used, the NetSentinel key must be connected to a computer running Windows 95 or Windows NT.

   - If the IPX/SPX protocol is used, *both* IPX *and* SPX must be loaded on the server computer and on each client computer running SLOPE/W.

2. The NetSentinel security key must be attached to the parallel printer port on one computer on the network.

   - This security computer may be the network file server or any one of the client computers.

3. The security computer must be running an appropriate version of the security server program.

   - Security server programs are included for Windows 95, Windows NT, Novell NetWare 3.x and 4.x, OS/2, and DOS.

   - The NetSentinel NLM security server is fully approved and certified by Novell Labs for use on NetWare file servers.

NOTE: If you are running the SLOPE/W Network Version using IPX/SPX under Windows 95, you must install Microsoft's NWLINK IPX software patch. This software patch fixes a bug in versions of Windows 95 prior to Service Release 2. You will be prompted to install this patch (if it is necessary) when you are running the SLOPE/W Setup program or the Network Software Setup program.

Installing the Rainbow Network Software

➢ **To install the Rainbow software:**

1. Choose Install Additional Software for Network Versions from the main Setup window.

   The Network Software Setup program begins execution.

2. Select the software components that you wish to install: the security servers, the security monitors, and the Sentinel system driver.

   - For the security servers, select the appropriate version for the operating system you are running on the security computer. You can select more than one security server version if you wish.

   - For the monitoring programs, select the appropriate versions for all client computers that you wish to run the security monitors. You can select more than one security monitor version.

   - Install the Sentinel system driver if you are also installing the Windows or OS/2 versions of the security server; for DOS or NetWare versions of the security server, you do not require a Sentinel system driver.

3. Follow the remaining Setup instructions.

   The security server and monitoring programs are copied to subdirectories within the specified directory. The subdirectories are named DOS, NW, OS2 and WIN32 (applicable to Windows 95 or NT) corresponding to the name of the operating systems. If you are installing server or monitoring programs for WIN32, Setup will create program folders for them. You can run these program by clicking on their icons.

➢ **To run the SLOPE/W Network Version:**

1. Attach the NetSentinel key to the security computer and run the installed security server.
2. Run the installed security monitoring program on any client computer on the network.
3. Run the installed SLOPE/W Network Version.

   The security monitoring program will indicate one SLOPE/W license in use.

Security Server Reference

Choosing the Security Computer

The GEO-SLOPE network software includes six versions of the security server program as shown in Table 2.2.

Table 2.2 Network Security Server Software Versions

Program	Description
NSRVDI.EXE	DOS IPX/SPX TSR server
NSRVDN.EXE	DOS NetBIOS TSR server
NSRVOM.EXE	Multi-protocol OS/2 server
NSRVNI.NLM	NetWare NLM IPX/SPX server
NSSRVICE.EXE	NetSentinel Service for Windows NT using IPX/SPX, NetBIOS, and TCP/IP
NSRVGX.EXE	Windows 32-bit IPX/SPX, NetBIOS, and TCP/IP server

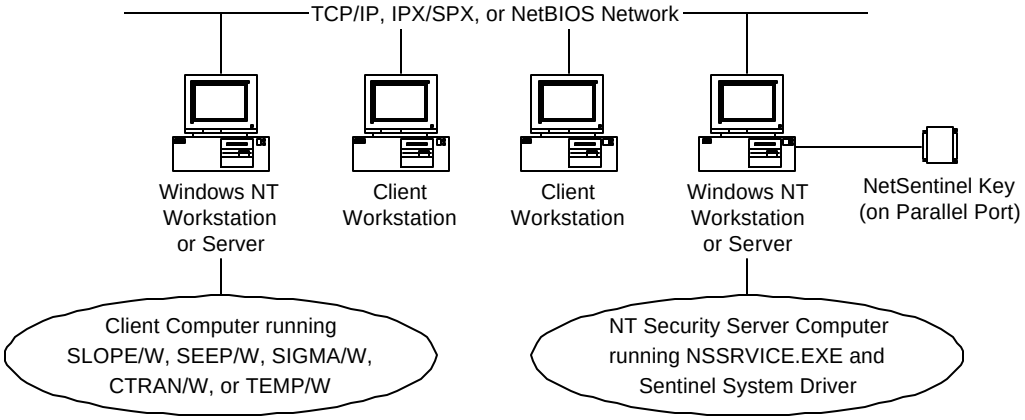
The different versions of the security server allow you the flexibility of running the program on a Windows 95, Windows NT, DOS or OS/2 workstation or on a Novell NetWare or Windows NT file server. The computer you choose will depend on your specific network environment and available computer resources. Remember that the NetSentinel security key must be connected to a parallel port on the security computer.

In most circumstances, the best option is to run a version of the security server on a computer that is always running. Your network file server, for example, provides the NetSentinel security server with a robust hardware platform; choosing a security computer that crashes frequently would force all SLOPE/W users to restart the program. The file server also provides a measure of physical security to the NetSentinel key, since the file server is normally locked in a limited-access facility.

Windows NT

If you are using Windows NT Server or Workstation on your network, the best option may to run NSSRVICE.EXE, the version of the NetSentinel security server that is implemented as a Windows NT service. Since it is an NT service, this security server is automatically started whenever the Windows NT operating system is started. There is no need to log on to Windows NT to start the security server, and the server will not be stopped when you log off from Windows NT. The NetSentinel Service supports NetBIOS, IPX/SPX, and TCP/IP.

The following diagram illustrates the NetSentinel security key and software connected to a typical Windows NT network:

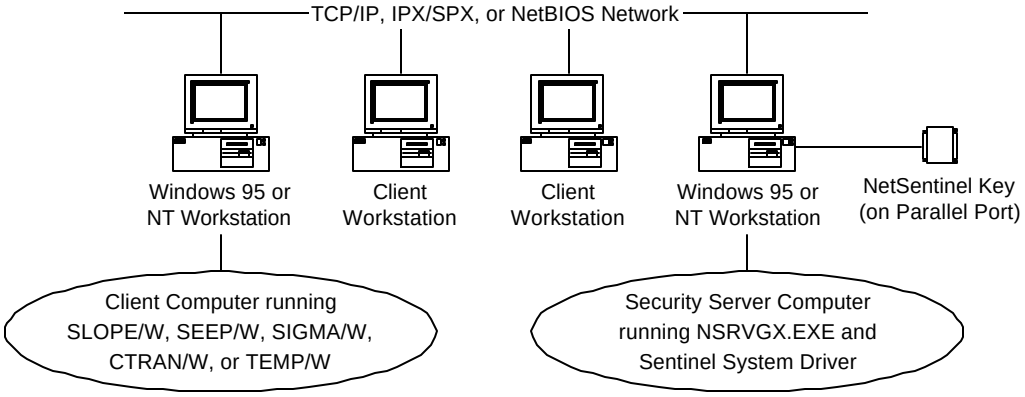

TCP/IP, IPX/SPX, or NetBIOS Network
Windows NT Workstation or Server
Client Workstation
Client Workstation
Windows NT Workstation or Server
NetSentinel Key (on Parallel Port)
Client Computer running SLOPE/W, SEEP/W, SIGMA/W, CTRAN/W, or TEMP/W
NT Security Server Computer running NSSRVICE.EXE and Sentinel System Driver

NetSentinel Security Key on a Windows NT network

Windows 95 or NT

The Win32 security server (NSRVGX.EXE) is a good choice if you are using Windows 95 or if you are using Windows NT and do not wish to use the NT Service security server. The Win32 security server is a native 32-bit Windows program and supports NetBIOS, IPX/SPX, and TCP/IP.

The following diagram illustrates the NetSentinel security key and software connected to a typical Windows 95 or NT network:

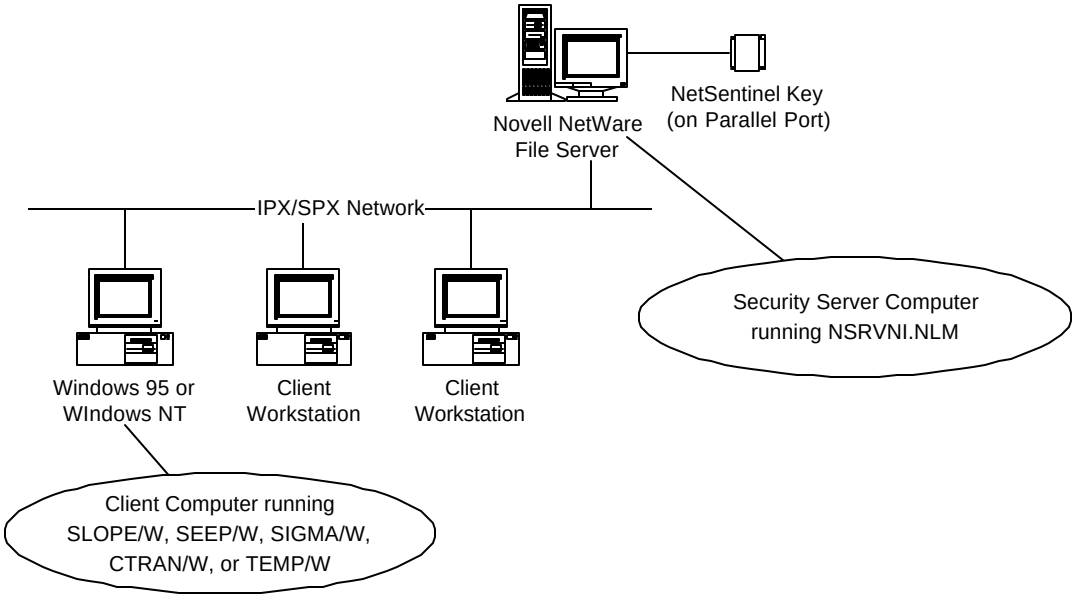

TCP/IP, IPX/SPX, or NetBIOS Network
Windows 95 or NT Workstation
Client Workstation
Client Workstation
Windows 95 or NT Workstation
NetSentinel Key (on Parallel Port)
Client Computer running SLOPE/W, SEEP/W, SIGMA/W, CTRAN/W, or TEMP/W
Security Server Computer running NSRVGX.EXE and Sentinel System Driver

NetSentinel Security Key on a Windows 95 or NT network

Novell NetWare

If you are using Novell NetWare, the best option may be to run the NLM (NetWare Loadable Module) version of the security server on the Novell file server. This NLM, tested and approved by Novell Labs, can be loaded and unloaded without rebooting the file server.

The following diagram illustrates the NetSentinel security key and software connected to a typical Novell NetWare network:

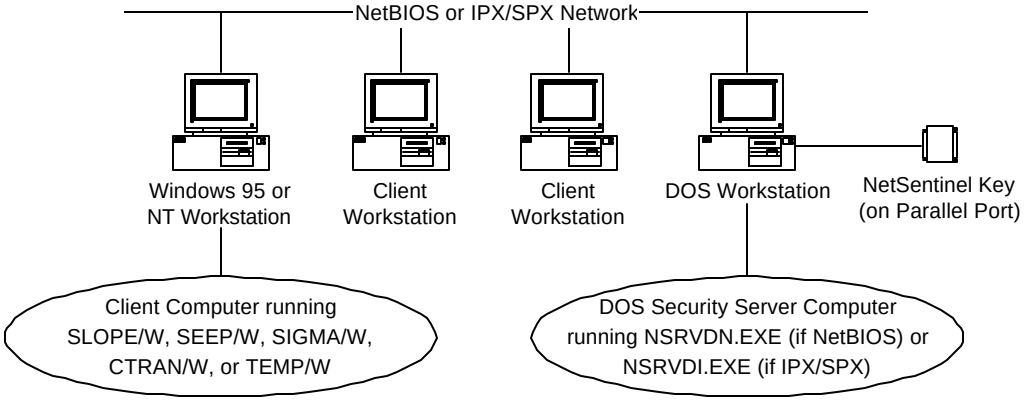


NetSentinel Security Key on a Novell NetWare Network

DOS

You might choose a DOS computer to be the security server if the computer is used infrequently. However, if you use the computer to run other programs and these programs crash, you may have to reboot your computer. This will restart the security server, forcing everyone using the SLOPE/W Network Version to restart SLOPE/W. It is therefore recommended that you choose one of the other server programs (e.g., the Win32 security server) instead of the DOS-based security servers. If you choose a DOS-based security computer, choose one that will remain up and running as much as possible, even if it is an older computer model.

The following diagram illustrates the NetSentinel security key and software connected to a typical NetBIOS network:



NetSentinel Security Key on a NetBIOS or IPX/SPX network

Running the Security Server on a NetWare File Server

The NetWare version of the security server runs as an NLM (NetWare Loadable Module) on a Novell NetWare 3.x or 4.x file server. This NLM, tested and approved by Novell Labs, can be loaded and unloaded without rebooting the file server.

➢ **To run the security server on a Novell NetWare NLM file server:**

1. Copy the file NSRVNI.NLM to your network file server. This file is installed by the Network Software Setup program and is located in the SERVER\NW sub-directory.

2. Attach the network security key to the parallel port on the file server.

3. Load the security server program from the command line into memory. For example, type:

   ```
   load nsrvni.nlm
   ```

   Since the NetWare version of the security server is a NetWare Loadable Module (NLM), you can also unload the program from the file server. For example, type:

   ```
   unload nsrvni.nlm
   ```

   The security server NLM will tell you if there are any licenses in use and give you the opportunity to change your mind before unloading.

NOTE: If you wish to load the security server automatically when the file server is booted, you can simply add the load command to the file server's AUTOEXEC.NCF file. Also, remember that the NLM version of the security server supports IPX/SPX clients only.

Table 2.3 shows the command line options supported by the NLM security server (the command line switches are not case sensitive).

Table 2.3 NLM Security Server Command Line Options

NLM Server Option	Description
/AT:<nnn>	Sets the timing delay in milliseconds between each instruction sent to the NetSentinel key. The default is auto-detection (/AT:0). In the case of a Novell file server running on a fast system, /AT:50 or /AT:100 is recommended. The default value of 0 causes the server to internally compute a value corresponding to a 10 microsecond wait (typically 80 on 486DX2 PC's). By increasing this number, more wait time can be given on computers for which an internally computed wait time is artificially low due to multi-level caches on the computer.
/DN:<name>	Changes the security server's department name from NETINEL to <name>. You do not need to use this option, since the SLOPE/W Network Version can only access a department name of NETINEL.
/DT:<nnn>	Sets the timing delay in milliseconds between establishing SPX connection and sending the handshake message. The default is 0 milliseconds. Specify /DT:50 if the SLOPE/W Network Version occasionally cannot find the NetSentinel key after it has been loaded and running for a while.
/H:<nnn>	Sets the maximum number of licenses that can be in use at any one time on this server to <nnn>. (The default is 150). Your effective license limit is the *smaller* of (1) the number you set here and (2) the sum of the limits of the keys connected to this server. Specifying a limit higher than what the attached keys support does not increase the license limit. Specifying a limit lower than what the attached keys support effectively disables some licenses.
/MS:<nnn>	Sets the maximum number of servers running on the network to <nnn>. The indicated value ranges from 1 to 10, and is used to determine the range of server names (e.g., NETINEL0, NETINEL1, etc.). If you are only using one security key, you do not need to use this option.
/N:<name>	Sets the name displayed by the security monitor program for this server to <name>. The default is your computer's Ethernet address (NetBIOS) or IPX node number (NetWare).
/P	Overrides the server's use of BIOS parallel port table addresses and uses the standard values 0x278, 0x378, and 0x3BC. This option is needed when the server is run on a machine where other software (such as PowerLAN) has zeroed out the BIOS table located in memory from 40:8 to 40:D.
/P:<port>	Overrides the server's use of BIOS parallel port table addresses and uses the hexadecimal address <port>. Up to three addresses may be specified. This option is needed when the server is run on a machine where other software (such as PowerLAN) has zeroed out the BIOS table located in memory from 40:8 to 40:D, and when a security key is located on a parallel port configured for an I/O location other than 0x278, 0x378, or 0x3BC. For example, **/p:278 /p:378** identifies parallel ports at I/O addresses 0x278 and 0x378.
/Q	Suppresses sign-on messages.
/S:<nnn>	Sets the maximum number of clients that can actively communicate with the server at one time to <nnn>. Note that half of the sessions are used to turn away clients. The default is 4 (two clients at a time).

/SL:<nnnn>	Sets the maximum number of sub-licenses expected to be open at any one time to <nnnn>. This option only applies if you are using multiple GEO-SLOPE products. The default is 256 product sub-licenses.
/ST	Enables strict license time-out enforcement. If this option is set, active licenses are immediately revoked and made available for reuse if the SLOPE/W Network Version has not communicated with the key for 20 minutes (This may happen if SLOPE/W crashes and is unable to free its license before it exits). Setting this option will automatically disconnect timed-out applications from the key. By default, a timed-out license is revoked only if another computer starts a GEO-SLOPE network version and there are no other licenses available (i.e., you've already reached your maximum user limit).
/W:<password>	Sets a password of up to 12 characters. If the server is set with a password option, that password will be required by the security monitoring program whenever licenses are being deleted. If the server is not set to require a password, the server will delete all licenses shown by the security monitor without requiring a password.
/?	Displays the available command line options and then terminates. Output can be redirected to a file using ">".

Running the NetSentinel Service Security Server under Windows NT

The Windows NT service version of the NetSentinel security server (NSSRVICE.EXE) supports IPX/SPX, NWLINK and TCP/IP protocols (For more information about TCP/IP support, see the Using TCP/IP with Windows 95 and NT section in the NetSentinel Configuration Reference in this chapter). Novell IPX/SPX client applications can communicate with the Win32 server if the NWLINK protocol is present on the workstation where the server is running.

NetBIOS and NetBEUI protocols are also supported. The NetSentinel Service supports whatever NetBIOS transports are installed under the NetBIOS interface. More than one NetBIOS may be present at the same time. The server supports Microsoft NetBEUI as well as NWLINK NetBIOS, which is interoperable with Novell NetBIOS clients.

NOTE: Before you can run the NetSentinel Service security server, you *must* install the NT Sentinel System Driver (Version 5.18 or later) to allow Windows NT to communicate with the NetSentinel key. If you are using an earlier version of the Sentinel driver, please install the latest version from the SLOPE/W CD-ROM; otherwise, Windows NT will generate an event log and the NT service will terminate.

The NetSentinel Service security server (NSSRVICE.EXE) can be run on a computer using Windows NT Server or Workstation 3.5 or higher. The Network Software Setup program installs NSSRVICE into the specified folder (e.g., \GSI_NET\NetServr\Win32).

- **To install the NetSentinel Service security server on a Windows NT computer:**

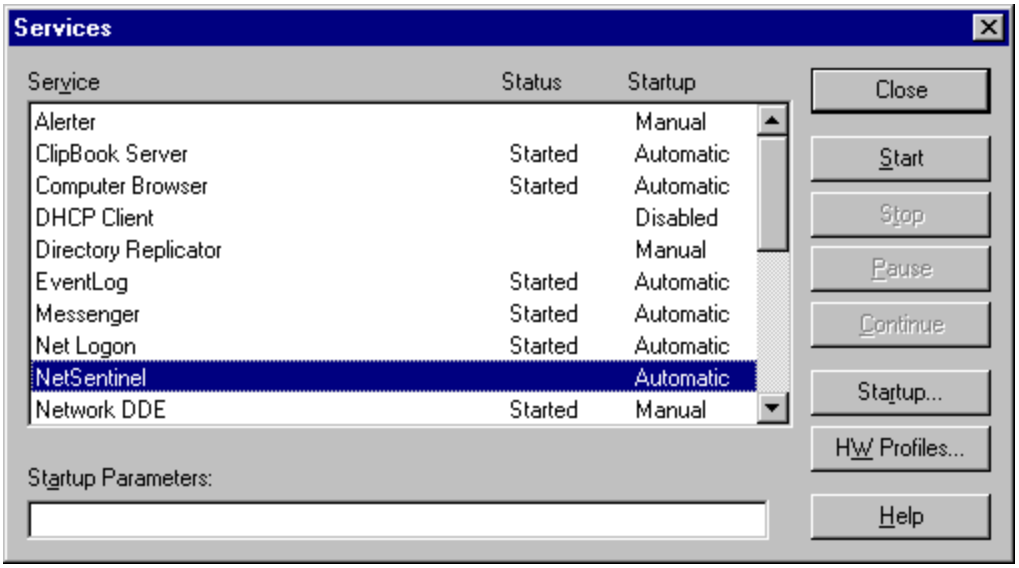
1. Attach the NetSentinel security key to the parallel port on the computer.
2. Install the NT Sentinel System Driver (Version 5.18 or later) to allow Windows NT to communicate with the NetSentinel key.
3. Copy NSSRVICE.EXE from the installed directory (e.g., \GSI_NET\NetServr\Win32) to the Windows NT %SystemRoot%\System32 directory (e.g., \WinNT\System32) on the computer.
4. To install the service, choose Run from the Start menu or Program Manager and run **NSSRVICE /I**

   The NetSentinel service will run automatically the next time you reboot your Windows NT computer. You do not need to log on to start the service.

A registry entry for the NetSentinel service is created under HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Services.

➢ **To run the NetSentinel Service security server:**

- Reboot your computer, or
- At the Windows NT command prompt, type NET START NETSENTINEL, or
- Run Control Panel and double-click on the Services applet. When the dialog box appears (as follows), select the NetSentinel Service and press the Start button.



NOTE: Status information pertaining to the NetSentinel service is reported to the Windows NT application log. Upon successful startup of the service, information such as server version, protocol stacks, and available keys is logged as two separate events in the application log. Please use the NT Event Viewer in the Administrator Tools group to view this information.

➢ **To un-install the NetSentinel Service security server:**

1. Log on to the Windows NT computer.
2. Stop the NetSentinel NT service by typing NET STOP NETSENTINEL at the NT command prompt. Alternatively, you can run Control Panel, double-click on the Services applet, select the NetSentinel Service, and press the Stop button.
3. Choose Run from the Start menu or Program Manager and run **NSSRVICE /U**

   The registry entry for the NetSentinel Service is removed from HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Services.

➢ **To display the version of the NetSentinel Service security server:**

- Choose Run from the Start menu or Program Manager and run **NSSRVICE /V**

NOTE: If you are using the NetBIOS or NETBEUI protocols, you must *only* use Lana Number 0. To view and modify this setting, run the Control Panel Network applet, choose Network Services, and select NetBIOS Interface. Click on the Properties button (or Configure button in NT 3.51) to display the current Lana Number settings.

Table 2.5 shows the command line options supported by the NetSentinel Service security server (the command line switches are not case sensitive). The specified options, if any, will take effect the next time the service is started.

Table 2.4 NetSentinel Service Security Server Command Line Options

NetSentinel Service Option	Description
/BI:<address>	Overrides the default "Find_Server" UDP broadcast address to direct the search over TCP/IP to a specified subnet. (The default address is set to 255.255.255.255, for a limited broadcast to all connected network segments).
/DN:<name>	Changes the security server's department name from NETINEL to <name>. You do not need to use this option, since GEO-SLOPE's network versions can only access a department name of NETINEL.
/H:<nnn>	Sets the maximum number of licenses that can be in use at any one time on this server to <nnn>. (The default is 150). Your effective license limit is the *smaller* of (1) the number you set here and (2) the sum of the limits of the keys connected to this server. Specifying a limit higher than what the attached keys support does not increase the license limit. Specifying a limit lower than what the attached keys support effectively disables some licenses.
/MS:<nnn>	Sets the maximum number of servers running on the network to <nnn>. The indicated value ranges from 1 to 10, and is used to determine the range of server names (e.g., NETINEL0, NETINEL1, etc.). If you are only using one security key, you do not need to use this option.
/N:<name>	Sets the name displayed by the security monitor program for this server to <name>. The default is your computer's Ethernet address (NetBIOS) or IPX node number (NetWare).
/RI:<num>	Defines the number of retry operations when searching for servers running over TCP/IP. The default is 3.
/SI: <nnnn>	Sets the number of threads devoted to handling TCP/IP clients to <nnnn>. Values range from 0 to 4; the default if 4. Specifying **/SN:0** disables all TCP/IP support.
/SL: <nnnn>	Defines the number of entries in the sub-license table.
/SN:<nnnn>	Sets the number of threads devoted to handling NetBIOS/NetBEUI clients to <nnnn>. Values range from 0 to 4; the default is 4. Specifying **/SN:0** disables all NetBIOS/NetBEUI support.
/ST	Enables strict license time-out enforcement. If this option is set, active licenses are immediately revoked and made available for reuse if the SLOPE/W Network Version has not communicated with the key for 20 minutes (This may happen if SLOPE/W crashes and is unable to free its license before it exits). Setting this option will automatically disconnect timed-out applications from the key. By default, a timed-out license is revoked only if another computer starts a GEO-SLOPE network version and there are no other licenses available (i.e., you've already reached your maximum user limit).
/SW:<nnnn>	Sets the number of threads devoted to handling IPX/SPX (NWLINK) clients to <nnnn>. Values range from 0 to 4; the default is 4. Specifying **/SW:0** disables all IPX/SPX (NWLINK) support.
/TI:<num>	Sets the time-out value in seconds for each retry operation when searching for servers running over TCP/IP. The default is 5.

/W:<password>	Sets a password of up to 12 characters. If the server is set with a password option, that password will be required by the security monitoring program whenever licenses are being deleted. If the server is not set to require a password, the server will delete all licenses shown by the security monitor without requiring a password.

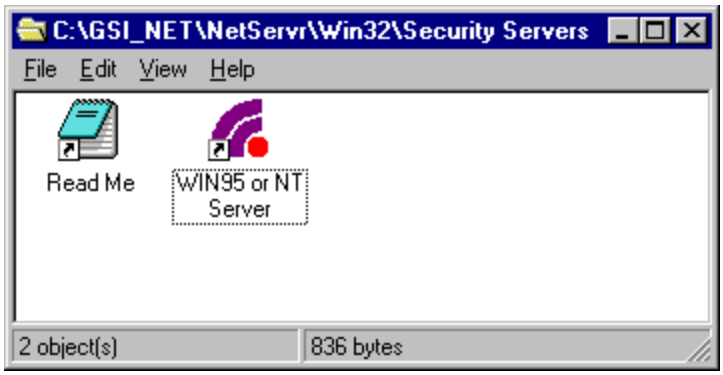
Running the Win32 Windows Security Server

The 32-bit Windows version of the NetSentinel security server (NSRVGX.EXE) supports IPX/SPX, NWLINK and TCP/IP protocols (For more information about TCP/IP support, see the Using TCP/IP with Windows 95 and NT section in the NetSentinel Configuration Reference in this chapter). Novell IPX/SPX client applications can communicate with the Win32 server if the NWLINK protocol is present on the workstation where the server is running.

NetBIOS and NetBEUI protocols are also supported. The Win32 server supports whatever NetBIOS transports are installed under the NetBIOS interface. More than one NetBIOS may be present at the same time. The server supports Microsoft NetBEUI as well as NWLINK NetBIOS, which is interoperable with Novell NetBIOS clients.

NOTE: Before you can run the Win32 security server under Windows NT, you *must* install the NT Sentinel System Driver to allow Windows NT to communicate with the NetSentinel key. If you are running under Windows 95, it is recommended that you install the Windows 95 Sentinel System Driver before running the Win32 security server.

The Win32 security server (NSRVGX.EXE) can be run on a Windows NT or Windows 95 computer. The Network Software Setup program installs NSRVGX and creates a folder containing NSRVGX for you:



➢ **To run the Win32 security server on a Windows NT or Windows 95 computer:**

1. Attach the NetSentinel security key to the parallel port on the computer.

2. Run NSRVGX from the folder created by the Network Software Setup program (e.g., \GSI_NET\NetServr\Win32\NSRVGX.EXE). Alternatively, you can run NSRVGX by choosing Run from the Start menu or Program Manager and specifying its full path.

   The server program displays a copyright message as it loads. When loaded, the program appears as an icon on the display screen. If you want to view information about the server as it runs, maximum its icon.

3. To unload the Win32 NetSentinel server, close the program.

NOTE: If you are using the NetBIOS or NETBEUI protocols, you must *only* use Lana Number 0. To view and modify this setting, run the Control Panel Network applet, choose Network Services, and select NetBIOS Interface. Click on the Properties button (or Configure button in NT 3.51) to display the current Lana Number settings.

Table 2.5 shows the command line options supported by the Win32 security server (the command line switches are not case sensitive).

Table 2.5 Win32 Security Server Command Line Options

Win32 Server Option	Description
/BI:<address>	Overrides the default "Find_Server" UDP broadcast address to direct the search over TCP/IP to a specified subnet. (The default address is set to 255.255.255.255, for a limited broadcast to all connected network segments).
/DN:<name>	Changes the security server's department name from NETINEL to <name>. You do not need to use this option, since GEO-SLOPE's network versions can only access a department name of NETINEL.
/H:<nnn>	Sets the maximum number of licenses that can be in use at any one time on this server to <nnn>. (The default is 150). Your effective license limit is the *smaller* of (1) the number you set here and (2) the sum of the limits of the keys connected to this server. Specifying a limit higher than what the attached keys support does not increase the license limit. Specifying a limit lower than what the attached keys support effectively disables some licenses.
/MS:<nnn>	Sets the maximum number of servers running on the network to <nnn>. The indicated value ranges from 1 to 10, and is used to determine the range of server names (e.g., NETINEL0, NETINEL1, etc.). If you are only using one security key, you do not need to use this option.
/N:<name>	Sets the name displayed by the security monitor program for this server to <name>. The default is your computer's Ethernet address (NetBIOS) or IPX node number (NetWare).
/Q	Suppresses sign-on messages.
/RI:<num>	Defines the number of retry operations when searching for servers running over TCP/IP. The default is 3.
/SI: <nnnn>	Sets the number of threads devoted to handling TCP/IP clients to <nnnn>. Values range from 0 to 4; the default if 4. Specifying **/SN:0** disables all TCP/IP support.
/SL: <nnnn>	Defines the number of entries in the sub-license table.
/SN:<nnnn>	Sets the number of threads devoted to handling NetBIOS/NetBEUI clients to <nnnn>. Values range from 0 to 4; the default is 4. Specifying **/SN:0** disables all NetBIOS/NetBEUI support.
/ST	Enables strict license time-out enforcement. If this option is set, active licenses are immediately revoked and made available for reuse if the SLOPE/W Network Version has not communicated with the key for 20 minutes (This may happen if SLOPE/W crashes and is unable to free its license before it exits). Setting this option will automatically disconnect timed-out applications from the key. By default, a timed-out license is revoked only if another computer starts a GEO-SLOPE network version and there are no other licenses available (i.e., you've already reached your maximum user limit).
/SW:<nnnn>	Sets the number of threads devoted to handling IPX/SPX (NWLINK) clients to <nnnn>. Values range from 0 to 4; the default is 4. Specifying **/SW:0** disables all IPX/SPX (NWLINK) support.
/TI:<num>	Sets the time-out value in seconds for each retry operation when searching for servers running over TCP/IP. The default is 5.

/W:<password>	Sets a password of up to 12 characters. If the server is set with a password option, that password will be required by the security monitoring program whenever licenses are being deleted. If the server is not set to require a password, the server will delete all licenses shown by the security monitor without requiring a password.

Running the Security Server on a DOS Computer

The DOS versions of the security server are provided in case you wish to run the security server on a rarely-used, DOS-based computer. Since DOS-based computers are frequently unstable and prone to crashing, it is recommended that you run one of the other security server programs (e.g., the Win32 security server) instead of the DOS-based security servers.

The DOS versions of the security server run as TSR (Terminate and Stay Resident) programs. One version uses the NetBIOS protocol, and the other version uses the IPX/SPX protocol.

➢ **To run the security server on a DOS computer:**

1. Determine whether your network uses the NetBIOS or IPX/SPX protocols.

   NetBIOS is a popular protocol supported by many networks, while IPX/SPX is the native Novell NetWare protocol.

2. If you are using NetBIOS, copy the files NSRVDN.EXE and NSRVDN.PIF to the local hard drive on the DOS server computer. These files are installed by the Network Software Setup program; their default location is in the \GSI_NET\NetServr\DOS directory.

3. Otherwise, if you are using IPX/SPX, copy the file NSRVDI.EXE to the local hard drive on the DOS server computer. This file is installed by the Network Software Setup program; its default location is in the \GSI_NET\NetServr\DOS directory.

4. Add a line to your AUTOEXEC.BAT file to run the security server each time the computer is turned on. For example, if you installed the NetBIOS security server, add the following line to AUTOEXEC.BAT:

   ```
   c:\nsrvdn
   ```

   If you installed the IPX/SPX security server, add the following line to AUTOEXEC.BAT:

   ```
   c:\nsrvdi
   ```

5. Attach the network security key to the parallel port of your computer and reboot the computer.

   The security server program is loaded into memory when the computer is restarted. This computer should always be running whenever anyone on the network is using the GEO-SLOPE network software.

If you are running Windows on the security computer, use the following guidelines :

- Always start the network transport protocol (IPX/SPX or NetBIOS) *and* the DOS-based security server *before* starting Windows.
- If you are using Windows for Workgroups on the security computer, add the following line to AUTOEXEC.BAT just before the security server is loaded:

  ```
  net start netbeui
  ```

This statement will start the NetBIOS protocol in real mode, which is required by the DOS version of the security server. Otherwise, Windows for Workgroups will load NetBIOS in 386 enhanced mode, and the security server will be unable to communicate properly with the security key

This statement should be added to AUTOEXEC.BAT for *all* computers running Windows for Workgroups and the GEO-SLOPE software, since the software must communicate with the security server in real mode when using NetBIOS.

- Never run an application in exclusive mode under Windows on the security computer. This will prevent the security server from communicating with the security key.

- If you are running DOS applications under Windows in 386 enhanced mode, make sure background processing is enabled. (This can be done by editing the application's PIF file and checking the **Background** option). Also, lock all memory used by the DOS application. (Edit the application's PIF file, select **Advanced**, and check **Lock Application Memory**). Not doing so may prevent the security server from communicating with the security key.

- Configure the parallel port to never warn about conflicts. (From the Windows Control Panel, select **386 Enhanced** and then select **Never Warn** for the appropriate port under **Device Contention**).

NOTE: Do not run one of the DOS-based security servers from a DOS box under Windows.

Table 2.6 shows the command line options supported by the DOS security server (the command line switches are not case sensitive).

Table 2.6 DOS Security Server Command Line Options

DOS Server Option	Description
/DN:<name>	Changes the security server's department name from NETINEL to <name>. You do not need to use this option, since GEO-SLOPE's network versions can only access a department name of NETINEL.
/DT:<nnn>	Sets the timing delay in milliseconds between establishing SPX connection and sending the handshake message. The default is 0 milliseconds. Specify /DT:50 if the SLOPE/W Network Version occasionally cannot find the NetSentinel key after it has been loaded and running for a while. This option applies to NSRVDI.EXE but not to NSRVDN.EXE.
/H:<nnn>	Sets the maximum number of licenses that can be in use at any one time on this server to <nnn>. (The default is 150). Your effective license limit is the *smaller* of (1) the number you set here and (2) the sum of the limits of the keys connected to this server. Specifying a limit higher than what the attached keys support does not increase the license limit. Specifying a limit lower than what the attached keys support effectively disables some licenses.
/MS:<nnn>	Sets the maximum number of servers running on the network to <nnn>. The indicated value ranges from 1 to 10, and is used to determine the range of server names (e.g., NETINEL0, NETINEL1, etc.). If you are only using one security key, you do not need to use this option.
/N:<name>	Sets the name displayed by the security monitor program for this server to <name>. The default is your computer's Ethernet address (NetBIOS) or IPX node number (NetWare).
/P	Overrides the server's use of BIOS parallel port table addresses and uses the standard values 0x278, 0x378, and 0x3BC. This option is needed when the server is run on a machine where other software (such as PowerLAN) has zeroed out the BIOS table located in memory from 40:8 to 40:D.
/P:<port>	Overrides the server's use of BIOS parallel port table addresses and uses the hexadecimal address <port>. Up to three addresses may be specified. This option is needed when the server is run on a machine where other software (such as PowerLAN) has zeroed out the BIOS table located in memory from 40:8 to 40:D, and when a security key is located on a parallel port configured for an I/O location other than 0x278, 0x378, or 0x3BC. For example, **/p:278 /p:378** identifies parallel ports at I/O addresses 0x278 and 0x378.
/Q	Suppresses sign-on messages.
/R	Conditionally unloads a previous instance of the server from memory, if and only if there are no open security sessions.
/S:<nnn>	Sets the maximum number of clients that can actively communicate with the server at one time. Note that half of the sessions are used to turn away clients. The default is 4 (two clients at a time).
/SL:<nnnn>	Defines the number of entries in the sub-license table.

/ST	Enables strict license time-out enforcement. If this option is set, active licenses are immediately revoked and made available for reuse if the SLOPE/W Network Version has not communicated with the key for 20 minutes (This may happen if SLOPE/W crashes and is unable to free its license before it exits). Setting this option will automatically disconnect timed-out applications from the key. By default, a timed-out license is revoked only if another computer starts a GEO-SLOPE network version and there are no other licenses available (i.e., you've already reached your maximum user limit).
/U	Unconditionally unloads a previous instance of the server from memory, whether or not there are open security sessions.
/W:<password>	Sets a password of up to 12 characters. If the server is set with a password option, that password will be required by the security monitoring program whenever licenses are being deleted. If the server is not set to require a password, the server will delete all licenses shown by the security monitor without requiring a password.
/?	Displays help information on the console and then terminates. Output can be redirected to a file using ">".

Running the Security Server on an OS/2 Computer

The OS/2 version of the security server runs as an OS/2 application. You can unload it by simply terminating the program, just as with any other OS/2 application.

NOTE: Before you can run the OS/2 security server, you *must* install the OS/2 Sentinel System Driver (SENTINEL.SYS) to allow OS/2 to communicate with the NetSentinel key. The location of this device driver must be specified in the CONFIG.SYS file. See the README.TXT file in the \SENTINEL\OS2 directory on the GEO-SLOPE distribution CD-ROM for information on installing the OS/2 Sentinel system driver.

➢ **To run the OS/2 security server:**

1. Install the OS/2 Sentinel System Driver to allow OS/2 to communicate with the NetSentinel key.
2. Add a line to your CONFIG.SYS file to load the OS/2 Sentinel device driver. For example, add the following line to CONFIG.SYS:

   ```
   DEVICE=C:\SENTINEL.SYS
   ```

3. Attach the NetSentinel security key to the parallel port on the computer.
4. Reboot the security computer.
5. Copy the file NSRVOM.EXE to the OS/2 security computer. This file is installed by the Network Software Setup program (run under Windows); its default location is in the C:\GSI_NET\NetServr\OS2 directory.
6. **For IBM or Microsoft Named Pipes:**

   Use the ACCESS CONTROL function in NET to share the pipe named \PIPE*deptname*, where *deptname* is your security server's department name. The default *deptname* is NETINEL.

   For Novell Named Pipes:

   Install the OS/2 Requester. Make the security server a Named Pipes server, and make every computer that is running the SLOPE/W Network Version a Named Pipes client.

7. Execute the security server program (NSRVOM) from the OS/2 command line, adding any options you desire.

8. Wait (about a minute) for the "Server Initialization completed" message to appear.

Once started, the server program appears as an icon on the screen. To stop running the server program, close the window in which the server program is running. To restart the server program, type **NSRVOM** followed by any desired options.

If desired, you can also run the program with no visible indication on the screen and no keyboard input/output. From the OS/2 command line or in the STARTUP.CMD file, type **DETACH NSRVOM** followed by any desired options. If you use this option, however, you will need to restart your computer if you wish to stop running the server program.

Table 2.7 shows the command line options supported by the OS/2 security server (the command line switches are not case sensitive).

Table 2.7 OS/2 Security Server Command Line Options

OS/2 Server Option	Description
/DN:<name>	Changes the security server's department name from NETINEL to <name>. You do not need to use this option, since GEO-SLOPE's network versions can only access a department name of NETINEL.
/H:<nnn>	Sets the maximum number of licenses that can be in use at any one time on this server to <nnn>. (The default is 150). Your effective license limit is the *smaller* of (1) the number you set here and (2) the sum of the limits of the keys connected to this server. Specifying a limit higher than what the attached keys support does not increase the license limit. Specifying a limit lower than what the attached keys support effectively disables some licenses.
/L:<type>:<dll>	Configures the server for a specific network operating system. By default, the server program looks for certain DLLs to determine which system is installed. Use the **/L** option if you have a network with multiple network operating systems installed and wish to control which is selected. Enter **/L:1** for IBM LAN Server (ACSNETB.DLL), or **/L:2** for Microsoft LAN Manager or Novell NetWare OS/2 Requester (NETAPI.DLL). If you want to specify the DLL to be used, enter the name after the number. For example, **/L:1:ACSNEW.DLL** loads the ACSNEW.DLL file and uses it as an IBM-type NetBIOS DLL.
/MS:<nnn>	Sets the maximum number of servers running on the network to <nnn>. The indicated value ranges from 1 to 10, and is used to determine the range of server names (e.g., NETINEL0, NETINEL1, etc.). If you are only using one security key, you do not need to use this option.
/N:<name>	Sets the name displayed by the security monitor program for this server to <name>. The default is your computer's Ethernet address (NetBIOS) or IPX node number (NetWare).
/Q	Suppresses sign-on messages.
/ SL: <nnnn>	Defines the number of entries in the sub-license table.
/SN:<nnn>	Sets the number of threads allocated for NetBIOS to <nnn>. In general, more threads provide better performance but require more memory. If you do not support NetBIOS, enter **/SN:0**. The default is 6.
/SP:<nnn>	Sets the number of threads allocated for Named Pipes to <nnn>. In general, more threads provide better performance but require more memory. If you do not support Named Pipes, enter **/SP:0**. The default is 6.
/ST	Enables strict license time-out enforcement. If this option is set, active licenses are immediately revoked and made available for reuse if the SLOPE/W Network Version has not communicated with the key for 20 minutes (This may happen if SLOPE/W crashes and is unable to free its license before it exits). Setting this option will automatically disconnect timed-out applications from the key. By default, a timed-out license is revoked only if another computer starts a GEO-SLOPE network version and there are no other licenses available (i.e., you've already reached your maximum user limit).

/W:<password>	Sets a password of up to 12 characters. If the server is set with a password option, that password will be required by the security monitoring program whenever licenses are being deleted. If the server is not set to require a password, the server will delete all licenses shown by the security monitor without requiring a password.
/?	Displays help information on the console and then terminates. Output can be redirected to a file using ">".

Security Monitor Reference

The Security Monitor Programs

The NetSentinel security monitoring program displays information about the security server and security key. This information includes server transport protocols, the number of licenses in use, the number of users who were disconnected after timing out, and the license limit for each key. You do not need to install the security monitor to use the SLOPE/W Network Version; however, the security monitor is useful for administrating the network software.

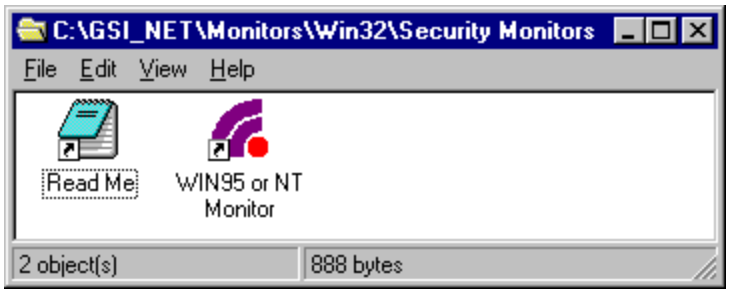
The security monitor can be run from any computer on the network. Before running the program, make sure you have started the appropriate network transport protocol (IPX/SPX, NetBIOS, Named Pipes, or TCP/IP).

The following versions of the security monitor programs are provided:

- ***WINMON*** A Windows-based tool that displays NetSentinel servers, keys, products, and users in the field. Unused licenses may be released and re-assigned from a single screen. The monitor may be customized by the system administrator to identify servers, users and products by name.
- ***DOSMON*** A DOS-based tool that displays all security servers, NetSentinel keys, and users on the network (except servers that use Named Pipes).
- ***OS2MON*** An OS/2-based tool that displays all security servers, NetSentinel keys, and users on the network.

Running WINMON, the Windows-Based Security Monitor

WINMON is the most flexible of the security monitoring programs. It is a 32-bit Windows program that can be run from any Windows 95 or Windows NT computer on the network. The Network Software Setup program installs WINMON and creates a folder containing WINMON for you:



➢ **To run WINMON on a Windows NT or Windows 95 computer:**

1. Run WINMON from the folder created by the Network Software Setup program. Alternatively, you can run WINMON from another network computer by choosing Run from the Start menu and specifying WINMON.EXE in the MONITORS\WIN32 sub-directory created by the Network Software Setup program.

   WINMON searches the network for NetSentinel security servers and NetSentinel keys and displays the names in the Server and Key drop-down edit boxes.

For all NetSentinel keys found on the network, the following information is displayed in the WINMON Keys list box:

- ***Name*** The name of the NetSentinel key (e.g., GEO-SLOPE)
- ***Type*** The type of the NetSentinel key (e.g., NS-C)
- ***AlgoID*** A hexadecimal number unique to each key name (e.g., 0000e3fd)
- ***Subs*** The number of sub-licenses (GEO-SLOPE products) controlled by the key. The GEO-SLOPE NetSentinel key controls five GEO-SLOPE software products, including SLOPE/W.
- ***Max*** The maximum number of licenses available (i.e., the maximum number of users that can run any GEO-SLOPE software simultaneously)
- ***Users*** The current number licenses in use (i.e., the current number of users running any GEO-SLOPE software)
- ***Peak*** The peak number of licenses used (i.e., the maximum number of users that have been running GEO-SLOPE software simultaneously)
- ***Locked*** **Yes** if the NetSentinel key is locked, **No** if it is unlocked and available for queries

If anyone is currently using any GEO-SLOPE software, the user names and last access time are displayed in the WINMON Users list box. The user name begins with **B:** or **S:** followed by the user's network address. **B:** refers to a base license and **S:** refers to a sub-license. A base license is granted the first time the user runs any of GEO-SLOPE's software products. The user is granted a sub-license for each GEO-SLOPE product running. For example, if a user starts running SLOPE/W DEFINE, a GEO-SLOPE base license and a SLOPE/W sub-license is granted. If the user then runs SLOPE/W DEFINE, a SLOPE/W sub-license is granted; a base license is not granted, since it was already given when SLOPE/W was started. If the user then runs SLOPE/W CONTOUR, no sub-licenses are granted, since the user already has a SLOPE/W sub-license.

2. In the Key drop-down list box, select GEO-SLOPE.

   Only GEO-SLOPE key information is displayed in the WINMON Key list box. The five GEO-SLOPE products are listed in the Product drop-down list box.

3. In the Product drop-down list box, select SLOPE/W.

   All SLOPE/W sub-licenses currently in use are displayed in the User list box.

4. Press the Edit Mapping File button if you wish to modify the names displayed by WINMON. The Edit Monitor Map dialog box appears.

5. To display actual user names in WINMON instead of network addresses, click the Add User button and enter the user information in the edit boxes.

   The WINMON names are stored in a file called MAPFILE.TXT in the same directory as WINMON.EXE. GEO-SLOPE has created names in this file for each GEO-SLOPE software product.

6. Choose Done in the Edit Monitor Map dialog box.

7. Choose Help Contents for more information on running WINMON.

8. To exit WINMON, press the Quit button.

WINMON can also be used to remove user licenses; this will disconnect the user from the NetSentinel key and cause the GEO-SLOPE application to terminate. Since you may not want everyone to delete licenses, you should start the security server with the password command line option. This will require the user to enter the password before the licenses can be deleted.

➢ **To remove licenses using WINMON:**

1. In the WINMON Users list box, select the licenses that you wish to delete.

   You can select a GEO-SLOPE license or a product sub-license. Deleting a GEO-SLOPE license will terminate all GEO-SLOPE applications currently running on the user's computer. Deleting a product sub-license (e.g., SLOPE/W) will only terminate SLOPE/W on the user's computer.

2. Press the Delete User License button.

   If the security server program was run with the /W password option, a password dialog box is displayed. Enter the password used to run the security server.

Running DOSMON, the DOS-Based Security Monitor

DOSMON can monitor NetSentinel keys attached to security computers running any NetBIOS or IPX/SPX based security server programs. DOSMON was the first security monitoring program available; it is not as full-featured or as easy to use as WINMON.

You can run DOSMON from DOS or from a DOS box under Windows. It is installed in the TOOLS\DOS sub-directory created by the Network Software Setup program.

When it is first started, DOSMON searches the network for all security keys; a bar graph is displayed, showing the progress of the search. Once the search is complete, a menu is displayed. Select **View by server** or **View by algorithm** to display information about the security key.

The following general rules should help you use DOSMON:

- To select an item from a list (and see more detailed data on it), move the highlight bar to it using the arrow keys and press ENTER.
- To return to the previous screen, press ESC. Pressing ESC from the first screen exits the program. The exit must be verified before the program will terminate.
- To update the date on your screen, press TAB. The message **Working - Please Wait** is displayed while new data is collected. If it is necessary to search the entire network, a bar graph is displayed, showing the progress of the update.
- For help on any screen, press the F1 key.
- The monitor will recognize a mouse. On menus, a single click moves the highlight bar, and a single click on the highlight bar or a double click on a non-highlighted selection selects the menu option. In lists, clicking on an item selects the item. Clicking on any bar at the bottom of the screen is the same as pressing the key highlighted in the bar.

In most cases, the monitor program will automatically detect the monitor type you are using. However, when a monochrome monitor is used with a color card, the monitor program will detect a color monitor. There may also be situations where a monochrome monitor is detected instead of a color monitor. To override automatic monitor type detection, set the environment variable PNLMONO to 1 for a monochrome monitor (i.e., type `SET PNLMONO=1`) or set PNLCOLOR to 1 for a color monitor (i.e., type `SET PNLCOLOR=1`). The environment variable must be set prior to starting the monitor program.

Table 2.8 shows the command line options supported by DOSMON (the command line switches are not case sensitive).

Table 2.8 DOSMON Command Line Options

DOSMON Option	Description
/DN:<name>	Changes the security server's department name from NETINEL to <name>. You do not need to use this option, since GEO-SLOPE's network versions can only access a department name of NETINEL.
/H, /?	Displays help information on the console and then terminates. Output can be redirected to a file using ">".
/I	Searches for security servers using the IPX/SPX protocol.
/MS:<nnn>	Sets the maximum number of servers running on the network to <nnn>. The indicated value ranges from 1 to 10, and is used to determine the range of server names (e.g., NETINEL0, NETINEL1, etc.). If you are only using one security key, you do not need to use this option.
/N	Searches for security servers using the NetBIOS protocol.
/S	Displays software security information (network license configuration and status) on the console, and then terminates. Output can be redirected to a file using ">". The information displayed is the same as that printed by **Print network data base**.

DOSMON will search only for those security servers that use the protocols specified on the command line. If no protocol is specified, the default is to use all protocols (e.g., **/I /P**).

Running OS2MON, the OS2-Based Security Monitor

OS2MON can monitor NetSentinel keys attached to security computers running any NetBIOS or IPX/SPX based security server programs. OS2MON is an OS/2-based version of DOSMON; it is not as full-featured or as easy to use as WINMON.

You can run OS2MON from the TOOLS\OS2 sub-directory created by the Network Software Setup program.

Table 2.9 shows the command line options supported by OS2MON (the command line switches are not case sensitive, and each option must be preceded by at least one space).

Table 2.9 OS2MON Command Line Options

OS2MON Option	Description
/A	Searches for security servers using the NetBIOS ACSNETB protocol.
/DN:<name>	Changes the security server's department name from NETINEL to <name>. You do not need to use this option, since GEO-SLOPE's network versions can only access a department name of NETINEL.
/H, /?	Displays help information on the console and then terminates. Output can be redirected to a file using ">".
/I	Searches for security servers using the IPX/SPX protocol.
/L:<type>:<dll>	Configures the monitor for a specific network operating system. By default, the monitor program looks for certain DLLs to determine which system is installed. Use the **/L** option if you have a network with multiple network operating systems installed and wish to control which is selected. Enter **/L:1** for IBM LAN Server (ACSNETB.DLL), or **/L:2** for Microsoft LAN Manager or Novell NetWare OS/2 Requester (NETAPI.DLL). If you want to specify the DLL to be used, enter the name after the number. For example, **/L:1:ACSNEW.DLL** loads the ACSNEW.DLL file and uses it as an IBM-type NetBIOS DLL.
/MS:<nnn>	Sets the maximum number of servers running on the network to <nnn>. The indicated value ranges from 1 to 10, and is used to determine the range of server names (e.g., NETINEL0, NETINEL1, etc.). If you are only using one security key, you do not need to use this option.
/N	Searches for security servers using the NetBIOS NETAPI protocol.
/O:<file>	Specifies the NETOEM (Network Named Pipes) DLL file name.
/P	Searches for security servers using the Named Pipes protocol.
/S	Displays software security information (network license configuration and status) on the console, and then terminates. Output can be redirected to a file using ">". The information displayed is the same as that printed by **Print network data base**.

OS2MON will search only for those security servers that use the protocols specified on the command line. If no protocol is specified, the default is to use all protocols (e.g., **/I /P /N /A**).

The default DLL file names are ACSNETB.DLL, NETAPI.DLL, and NETOEM.DLL. Your LIBPATH is used to locate these DLLs. If you specify a DLL file name without a path and extension (for example, NEWDLL) the monitor uses your LIBPATH to locate the file. If the specified DLL cannot be located, the associated protocol will not be supported. An error message is displayed only if no transport protocols are available.

The following are examples of running OS2MON:

- ***OS2MON*** Searches for security servers using IPX/SPX, Named Pipes, NetBIOS NETAPI, and NetBIOS ACSNETB. This is the default.
- ***OS2MON /I /P*** Searches for security servers using IPX/SPX and Named Pipes.

- ***OS2MON /L:1:NEWDLL*** The ACSNETB DLL will be NEWDLL.DLL and must be located in a LIBPATH directory.

- ***OS2MON /O:C:\NETWARE\NEWDLL.DLL*** The NETOEM DLL will be C:\NETWARE\NEWDLL.DLL.

NetSentinel Configuration Reference

Banyan Vines

Environments Supported

NetSentinel Server OS	Protocol	Server Module
DOS	NetBIOS	NSRVDN.EXE
Windows NT, Windows 95	NetBIOS	NSRVGX.EXE

Configuration Issues

If the key is not seen by a particular Banyan VINES client, you may need to increase the number of NetBIOS sessions and commands that are being allocated for that client station (minimum 8 sessions and 12 commands). Also, a NetBIOS name may need to be created on the Banyan server and the NetBIOS software support must be installed.

The following are the minimum requirements for configuring your Banyan network to allow successful execution of NetBIOS application:

- Log into a Banyan VINES server from any station sharing or accessing a NetSentinel key. The server must have previously created a NetBIOS name, using the MSERVICE utility.

- On each client using the NetBIOS name, the *AUTOEXEC.BAT* must be modified. Add a line after the BAN statement stating:

  SETNETB <NetBIOS service name>

- NetBIOS software support must be enabled on every client sharing or accessing the NetSentinel key. Run PCCONFIG.EXE, and select ***3 - Special Software Settings***. Choose ***1 - Load Resident NetBIOS Emulation software***, and set it to ***Y*** (Yes).

- If you are running NSRVGX.EXE on Windows NT, you must install the Sentinel System Driver for Windows NT before starting the server.

- To increase the number of NetBIOS sessions and commands, run the PCCONFIG.EXE program. Choose ***2 - Login Environment Options***. Choose ***5 - Set Maximum NetBIOS Sessions*** and set the value. On the system acting as the NetSentinel server, the minimum session value is 8. Press F10 to save and ESC to return to the Login Environment Options menu. Choose ***6 - Set Maximum NetBIOS Commands*** and set the value.

  On the system acting as the NetSentinel server, the minimum command value is 12.

Known Problems

NetSentinel has been designed to work on any standard NetBIOS implementation. However, it has been discovered that Banyan VINES' implementation of NetBIOS has problems that, under some circumstances, will cause the workstation where the security server is running to fail.

In addition, Banyan VINES versions 4.10 and later contain a problem that causes the VINES NetBIOS emulator to fail, displaying the message: "Fatal NetBIOS Emulation Error", or causing the workstation running the NetSentinel client or server to hang.

The Banyan error message is, according to Banyan, an indication that the workstation's memory has been corrupted.

Banyan VINES has officially notified Rainbow Technologies that they have resolved this NetBIOS problem. The fix for this problem is included in v5.53.6 of Banyan VINES, and is available to v5.52.5 users as user-installable patch DD-1.

If you wish to obtain the v5.53.6 update or DD-1 patch, you should contact your Banyan VINES reseller for assistance.

IBM LAN Server/Requester 2.x and 3.x

Environments Supported

NetSentinel Server OS	Protocol	Server Module
DOS	NetBIOS	NSRVDN.EXE
OS/2	NetBIOS, Named Pipes	NSRVOM.EXE

Configuration Issues

On all systems acting as NetSentinel servers and clients, you should determine whether NetBIOS components of more than one network operating system are present. It is possible for components of two network operating systems to be present on the same computer.

For example, the IBM Communication Manager may be installed to provide asynchronous communications support, while LAN Manager is installed for LAN services. In this case, both ACSNETB.DLL (IBM NetBIOS) and NETAPI.DLL (IBM, Microsoft, and Novell) may be resident. This could cause a failure to properly initialize NetBIOS during server startup if the NetBIOS interface DLL used is not the correct DLL for the installed NetBIOS protocol stack.

To avoid possible problems, the OS/2 security server (NSRVOM.EXE) should be started with one of the following command-line parameters:

/L:1 ACSNETB.DLL support for IBM LAN Server

/L:2 NETAPI.DLL support for Microsoft LAN Manager and Novell NetWare

In the case where the system has two different vendor's versions of NETAPI.DLL installed, such as IBM LAN Manager and Novell NetWare, the OS/2 security server should be started with the appropriate switch and the path to the specific DLL to be used. For example,

```
NSRVOM /L:2 C:\NETWARE\NETAPI.DLL
```

To ensure that there will be an adequate supply of NetBIOS resources to support the NetSentinel Server(s) and Client(s), perform the following with the IBM LAN Server/Requester software:

- Configure your NetBIOS resources by selecting the Install/Config folder and then pressing the Advanced button.
- Select the Configure a Component option from the pop-up menu.
- Choose Adapter from the next menu, and you will be able to edit the fields that control the number of sessions, commands (NCBS), and NetBIOS names. Increase these as needed.

Under IBM LAN Server, a lack of resources can cause the NetSentinel server to fail to start NetBIOS. In some cases, the NetSentinel server runs; however, there are no resources remaining for the OS/2 Monitor (OS2MON.EXE). The workstation executes the program, but does not find the NetSentinel. Performing the following could help:

- Start the NetSentinel server using a small number of threads allocated for NetBIOS, such as:

  ```
  NSRVOM /SN:1
  ```

- Increase the Sessions, Commands, and/or NetBIOS names as described above. The changes can also be made manually by modifying the PROTOCOL.INI file as follows:

  ```
  Sessions=100
  NCBS=100
  ```

The resources available for this workstation is the difference between the values in PROTOCOL.INI and IBMLAN.INI. For example, if PROTOCOL.INI is set for Sessions=100, and IBMLAN.INI is set for Sessions=40; then there are 60 sessions left for NetSentinel and other NetBIOS applications.

If you are using the OS/2 security server (NSRVOM.EXE), verify it loads completely. The message "NetSentinel Start-Up Complete" will appear.

In addition, do not Close or CTRL-C this process or the server will be stopped. However, minimizing does not stop the server.

LANTastic

Environments Supported

NetSentinel Server OS	Protocol	Server Module
DOS	NetBIOS	NSRVDN.EXE

Configuration Issues

In a LANTastic network, access to the NetSentinel server can require up to two minutes when the NetBIOS client application is loaded on the same node as the NetSentinel security server.

Artisoft has released a new version of their NetBIOS driver for LANTastic that eliminates this problem. A copy of this new driver is available on the Artisoft bulletin board system.

Artisoft BBS telephone number: (602) 884-8646

Communication parameters: 96, N, 8, 1

NetBIOS driver file: contact Artisoft for the file name

Microsoft LAN Server/Requester 2.0 and 2.1

Environments Supported

NetSentinel Server OS	Protocol	Server Module
DOS	NetBIOS	NSRVDN.EXE
Windows NT, Windows 95	NetBIOS	NSRVGX.EXE
OS/2	NetBIOS, Named Pipes	NSRVOM.EXE

Configuration Issues

On all systems acting as NetSentinel servers and clients, you should determine whether NetBIOS components of more than one network operating system are present. It is possible for components of two network operating systems to be present on the same computer.

For example, the IBM Communication Manager may be installed to provide asynchronous communications support, while LAN Manager is installed for LAN services. In this case, both ACSNETB.DLL (IBM NetBIOS) and NETAPI.DLL (IBM, Microsoft, and Novell) may be resident. This could cause a failure to properly initialize NetBIOS during server startup if the NetBIOS interface DLL used is not the correct DLL for the installed NetBIOS protocol stack.

To avoid possible problems, the OS/2 security server (NSRVOM.EXE) should be started with one of the following command-line parameters:

/L:1 ACSNETB.DLL support for IBM LAN Server

/L:2 NETAPI.DLL support for Microsoft LAN Manager and Novell NetWare

In the case where the system has two different vendor's versions of NETAPI.DLL installed, such as IBM LAN Manager and Novell NetWare, the OS/2 security server should be started with the appropriate switch and the path to the specific DLL to be used. For example,

```
NSRVOM /L:2 C:\NETWARE\NETAPI.DLL
```

To ensure that there will be an adequate supply of NetBIOS resources to support the NetSentinel Server(s) and Client(s), perform one of the following:

- Manually edit the PROTOCOL.INI file. You will find the NetBIOS parameters under the heading:

  ```
  [NetBEUI_XIF]
  ```

  The LAN Manager installation software does not place entries in the PROTOCOL.INI file for any NetBIOS parameters that are at default. You will need a copy of the LAN Manager documentation while editing this file.

  Then, increase the NetBIOS sessions, commands, and/or maximum names.

- Or, run the LAPS utility to increase the maximum allowable value for any of the NetBIOS parameters mentioned above.

Verify that each client wishing to execute the application has started LAN Manager workstation services. To do so, run NET START WORKSTATION.

If you are running NSRVGX.EXE on Windows NT, you must install the Sentinel System Driver for Windows NT before starting the server.

If you are using the OS/2 security server (NSRVOM.EXE), verify it loads completely. The message "NetSentinel Start-Up Complete" will appear.

In addition, do not Close or CTRL-C this process or the server will be stopped. However, minimizing does not stop the server.

Novell NetWare 3.x and 4.x

Environments Supported

NetSentinel Server OS	Protocol	Server Module
DOS	IPX/SPX	NSRVDI.EXE
DOS	NetBIOS	NSRVDN.EXE
NetWare	IPX/SPX	NSRVNI.NLM
Windows NT	IPX/SPX, NetBIOS	NSRVGX.EXE
Windows 95	IPX/SPX, NetBIOS	NSRVGX.EXE
OS/2	NetBIOS, Named Pipes	NSVROM.EXE

NOTE: If you are using the IPX/SPX protocol under Windows 95, you must install Microsoft's NWLINK IPX software patch. You will be prompted to install this patch when you are running the SLOPE/W Setup program or the Network Software Setup program.

Configuration Issues

Current versions of the NetWare DOS and Windows drivers should be used. The DOS drivers are available from Novell, CompuServe (in the files VLMUP3.EXE and NET33X.EXE); or your network card manufacturer. The minimum version required for IPX.COM is at least 3.10, IPXODI.COM is at least 1.20, and NETX.EXE is at least 3.26.

If the GEO-SLOPE software cannot find the NetSentinel key and returns an error message of –19, then you should update each NetSentinel server and client workstation with the above mentioned drivers.

If ODI drivers are utilized, verify IPXODI.COM is not being started with any arguments. Specifically, issuing the command, IPXODI /A, disables SPX services and will prevent the GEO-SLOPE software from successfully accessing the NetSentinel.

You may use PSERVER to share a printer that is connected to a port with a shared NetSentinel key, but PSERVER must not use interrupts. To not use interrupts, run PSERVER <printserver>, and in the configuration stating Use Interrupts? (Y/N), select N for "No", then save the configuration.

Windows for Workgroups 3.11 (NetBEUI)

Environments Supported

NetSentinel Server OS	Protocol	Server Module
DOS	NetBIOS	NSRVDN.EXE

Configuration Issues

Attaching the NetSentinel security key to a computer running Windows for Workgroups requires you to run the NSRVDN.EXE security server in DOS before starting Windows. To do so, you must add one of the following statements to your AUTOEXEC.BAT file:

- ***NET START NETBEUI*** Workstation can share its files and printers, but cannot connect to other shared files and printers.
- ***NET START FULL*** or ***NET START WORKSTATION*** Workstation cannot share its files and printer, but can connect to other shared files and printers.
- ***NET START BASIC*** Workstation cannot share its files and printers. It can connect to shared files and directories using the NET USE command at the DOS prompt outside Windows for Workgroups.

Due to the above limitations, it is highly recommended that you run the NetSentinel security server on a computer other than the Windows for Workgroups server.

The "IPX/SPX Compatible Transport with NetBIOS" or "IPX/SPX Compatible Transport" protocol must NOT be loaded. The system will hang when the NetSentinel application checks the IPX/SPX for a NetSentinel key.

To eliminate these limitations, an alternative approach is to execute the NetSentinel server (NSRVDN.EXE) in a Windows for Workgroups DOS box.

➢ **To run NSRVDN.EXE in a Windows for Workgroups DOS box:**

1. Start Windows for Workgroups.
2. Run NSRVDN.PIF to create a DOS box.

   NSRVDN.PIF is located in the same directory as NSRVDN.EXE. It contains the appropriate settings for running NSRVDN.EXE under Windows for Workgroups.

3. Run NSRVDN.EXE inside the DOS box.

NOTE: SLOPE/W Version 4 (or higher) *cannot* be run under Windows for Workgroups; SLOPE/W can only be run under Windows 95 or Windows NT. Only the security server and security monitor programs can be run under Windows for Workgroups.

Windows NT / Windows NT with Novell NetWare

Environments Supported

NetSentinel Server OS	Protocol	Server Module
Windows NT	NWLink/IPX, NetBIOS, TCP/IP	NSRVGX.EXE
Windows NT	NWLink/IPX, NetBIOS, TCP/IP	NSSRVICE.EXE

Configuration Issues

The Windows NT 32-bit server can be run as an Application (NSRVGX.EXE) or as an NT Service (NSSRVICE.EXE). The advantage of running the NT Service is that NSSRVICE.EXE is automatically started whenever the Windows NT operating system is started. There is no need to log on to Windows NT to start the security server, and the server will not be stopped when you log off from Windows NT. For information on installing NSSRVICE.EXE, see the Running the NetSentinel Service Security Server under Windows NT section in this chapter.

If you wish to run NSRVGX.EXE, however, and have it automatically load during booting, you can configure your system in the following way:

1. Add the value AutoAdminLogon: DWORD:1 to the following key in the Registry Editor by using REGEDT32.EXE:

   HKEY_LOCAL_MACHINE - SOFTWARE - Microsoft - Windows NT - Current Version - Winlogon

   To add the value from the Settings/Control Panel/Network Icon, select Edit, New, DWORD value.

2. Verify the value for DefaultUserName and DefaultPassword. To verify from the Settings/Control Panel/Network Icon, select Edit, New, String Value. If there are no values, add them.

   This will automatically log the user in. Place the NSRVGX.EXE in the STARTUP folder and the system should automatically load the server on every re-boot.

If the GEO-SLOPE application and the Win32 NetSentinel server are running on the same workstation, then the TASKING option of CONTROL PANEL's SYSTEM applet must be configured as "Foreground and Background Applications Equally Responsive" to avoid task scheduling problems.

The Windows NT Sentinel System Driver must be installed on the computer running the Win32 security server program. This driver is installed from the GEO-SLOPE distribution CD-ROM.

NOTE: For information about TCP/IP support, see the Using TCP/IP with Windows 95 and NT section in this chapter.

Known Problems

Due to an incompatibility between NWLink/IPX's router packet definition and Novell's routing software, it is impossible at this time to access a NetSentinel server across a Novell router. Novell and Microsoft are currently investigating this issue.

Windows 95 / Windows 95 with Novell NetWare

Environments Supported

NetSentinel Server OS	Protocol	Server Module
Windows 95	NWLink/IPX, NetBIOS, TCP/IP	NSRVGX.EXE

Configuration Issues

Currently, the Windows 95 server only *officially* supports the NetBIOS/NetBEUI and TCP/IP protocols.

Rainbow Technologies has implemented IPX/SPX support in the Windows 95 server, and is actively working with Microsoft to overcome issues with Windows 95, multi-threaded applications, and IPX/SPX. Recently, Microsoft released a Windows 95 Beta patch, NWLNKUPD.EXE, that seems to correct these issues. You are prompted to install this patch when you install the GEO-SLOPE application software from the distribution CD-ROM. This patch also seems to resolve issues with multi-threaded applications (like the NetSentinel client) running on Windows 95 utilizing the IPX/SPX protocol.

Installation of Novell's Windows 95 drivers seems to help as well, but does not completely resolve all the issues. We recommend installation of the NWLNKUPD.EXE patch instead.

NOTE: Unlike Windows NT, the Sentinel System Driver does *not* have to be loaded to run the Windows 95 server (NSRVGX.EXE).

To use the Windows 95 security server with the TCP/IP protocol, the TCP/IP protocol must be installed on the workstation acting as the NetSentinel server. From the Settings/Control Panel/Network Icon, verify that TCP/IP is listed as a protocol. If not, select Add, highlight Microsoft, add TCP/IP and click OK. A Standard Windows 95 installation does not install the TCP/IP Protocol stack by default. You must install this service.

NOTE: For information about TCP/IP support, see the Using TCP/IP with Windows 95 and NT section in this chapter.

Using TCP/IP with Windows 95 and NT

SLOPE/W Version 4 can use the TCP/IP protocol to connect to one of the Win32 NetSentinel security servers (i.e., NSRVGX.EXE or NSSRVICE.EXE). You will need to install TCP/IP on both the client computer and the security server computer. If you are using a non-Win32 security server, such as NSRVDN.EXE or NSRVDI.EXE, you cannot use TCP/IP to find the security server; instead, you will need to install additional protocols, such as NetBIOS or IPX/SPX.

By default, when SLOPE/W uses TCP/IP, it will only search for the NetSentinel security server on its local subnet. Your *local subnet* is defined by masking your computer's IP address with its subnet mask. For example, if your computer has an IP address of 192.9.100.1 and a subnet mask of 255.255.255.0, the local subnet will consist of all addresses starting with 192.9.100. If you designate your computer as the NetSentinel security server, SLOPE/W can be run from any other computer on this local subnet (e.g., from a computer with an IP address of 192.9.100.7 and a subnet mask of 255.255.255.0).

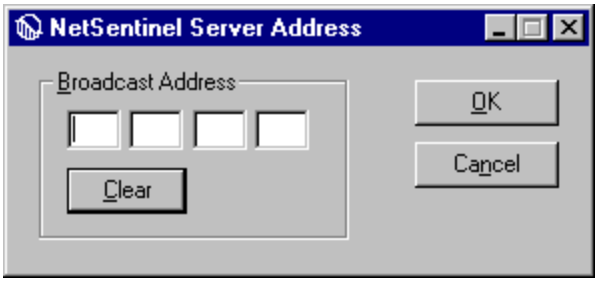
This default configuration is appropriate for most networks, which typically have the security server and the client computers on the same network segment. However, if one of your client computers is on a different subnet, you will not be able to run SLOPE/W using the default TCP/IP settings. For example, if your client computer has an IP address of 192.9.101.7 and a subnet mask of 255.255.255.0, its subnet is 192.9.101; it will not be able to find the NetSentinel security server, since it is on a different subnet.

In this case, you must override the default TCP/IP settings and specify either a *unicast address* or a *directed broadcast* on the client computer. A *unicast address* is simply the IP address of the NetSentinel security server. If you specify a unicast address, SLOPE/W will only check this IP address when it looks for the security server. You should use this option if you know that the IP address of the security server will not change.

If you specify a *directed broadcast* on the client computer, SLOPE/W will search for the NetSentinel security server on a particular subnet. A directed broadcast consists of the subnet address followed by all 1's (in binary). For example, if you wish to search for the security server on the 192.9.100 subnet, you should specify a directed broadcast address of 192.9.100.255. The advantage of using a directed broadcast is that you can move the security server to any computer on the specified subnet; it will still be found by SLOPE/W, because you are specifying the subnet to search, not the exact IP address. In this example, the security server could be any computer on the 192.9.100 subnet, such as 192.9.100.7 or 192.9.100.145.

➢ **To specify a unicast address on the client computer:**

1. On the client computer, run **\OfficeV4\Network\Tcpip\ServAddr.exe** from the GEO-SLOPE Office CD-ROM. The following window appears:



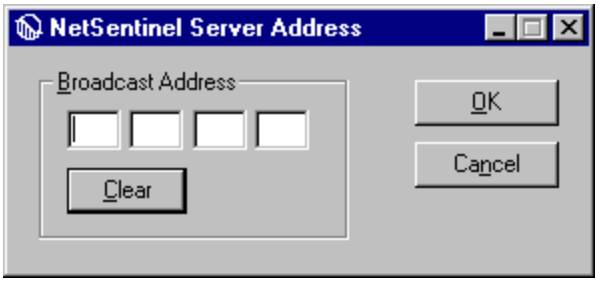
2. In the Broadcast Address edit boxes, type the IP address of the NetSentinel security server computer.

3. Select OK. The broadcast address is stored in the Windows Registry under the keyword \HKEY_LOCAL_MACHINE\SOFTWARE\GEO-SLOPE\MRUSystem\NetSentinel-TCPIP-BroadcastAddr.

   When you run SLOPE/W, it will look for the security server at this IP address.

➢ **To specify a directed broadcast on the client computer:**

1. On the client computer, run **\OfficeV4\Network\Tcpip\ServAddr.exe** from the GEO-SLOPE Office CD-ROM. The following window appears:



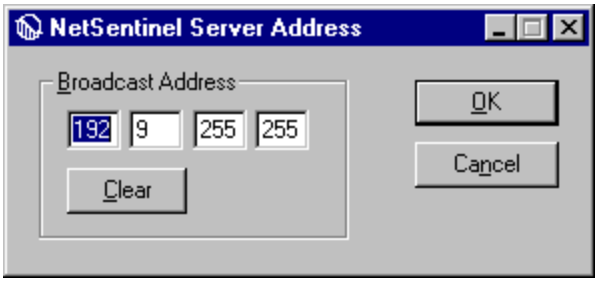
2. Type the broadcast address that you want SLOPE/W to use when it searches for the NetSentinel security server. For example, to search on the 192.18 subnet, type 192.18.255.255 as the broadcast address.

3. Select OK. The broadcast address is stored in the Windows Registry under the keyword \HKEY_LOCAL_MACHINE\SOFTWARE\GEO-SLOPE\MRUSystem\NetSentinel-TCPIP-BroadcastAddr.

   When you run SLOPE/W, it will perform a directed broadcast to this subnet in order to find the security server.

➢ **To return to the default TCP/IP settings (i.e., to broadcast to the local subnet):**

1. On the client computer, run **\OfficeV4\Network\Tcpip\ServAddr.exe** from the GEO-SLOPE Office CD-ROM. The following window appears, containing the current broadcast address:



2. Select the Clear button to remove the broadcast address.

3. Select OK to return to the default TCP/IP settings. The broadcast address keyword \HKEY_LOCAL_MACHINE\SOFTWARE\GEO-SLOPE\MRUSystem\NetSentinel-TCPIP-BroadcastAddr is deleted from the Windows Registry.

   When you run SLOPE/W, it will perform a broadcast to its local subnet in order to find the security server.

Chapter 3 SLOPE/W Tutorial

An Example Problem **3**

Defining the Problem **4**
- Set the Working Area 4
- Set the Scale 5
- Set the Grid Spacing 6
- Saving the Problem 6
- Sketch the Problem 7
- Specify the Analysis Methods 9
- Specify the Analysis Options 10
- Define Soil Properties 10
- Draw Lines 12
- Draw Piezometric Lines 14
- Draw the Slip Surface Radius 16
- Draw the Slip Surface Grid 17
- View Preferences 19
- Sketch Axes 21
- Display Soil Properties 23
- Label the Soils 26
- Add a Problem Identification Label 30
- Verify the Problem 33
- Save the Problem 35

Solving the Problem **36**
- Start Solving 36
- Quit SOLVE 37

Viewing the Results **38**
- Draw Selected Slip Surfaces 39
- View Method 40
- View the Slice Forces 41
- Draw the Contours 42
- Draw the Contour Labels 43
- Plot a Graph of the Results 44
- Print the Drawing 47

Using Advanced Features of SLOPE/W **49**
- Specify a Rigorous Method of Analysis 49
- Perform a Probabilistic Analysis 50
- Import a Picture 60

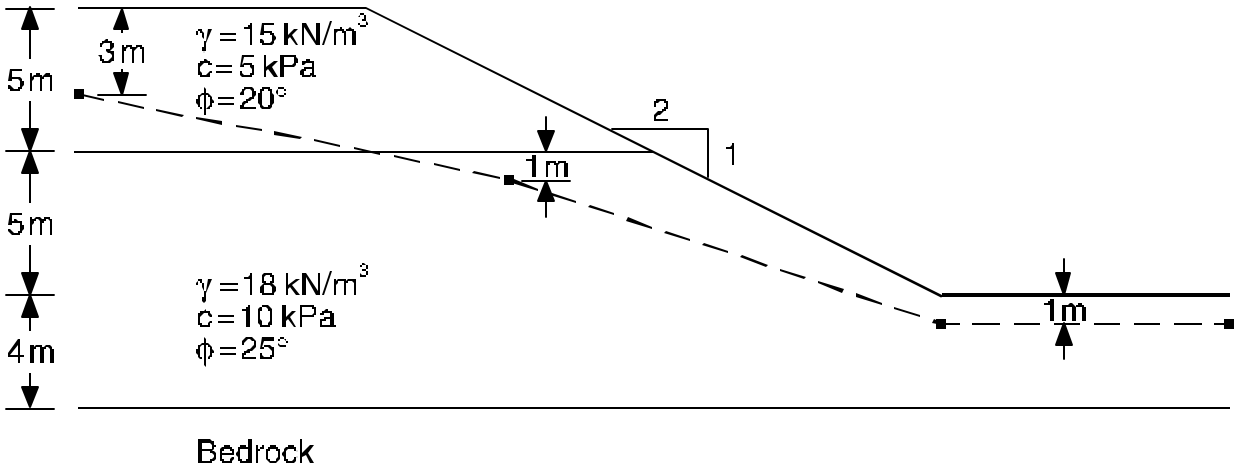
An Example Problem

This chapter introduces you to SLOPE/W by presenting the step-by-step procedures involved in analyzing a simple slope stability problem. By executing each step in the sequence presented, you will be able to define a problem, compute the factors of safety, and view the results. By completing this exercise, you can quickly obtain an overall understanding of the features and operations of SLOPE/W.

Figure 3.1 presents a schematic diagram of a slope stability problem. The objective is to compute the minimum factor of safety and locate the critical slip surface location.

The slope is cut in two materials at 2:1 (horizontal:vertical). The upper layer is 5 m thick and the total height of the cut is 10 m. Bedrock exists 4 m below the base of the cut. The pore-water pressure conditions are depicted by the piezometric line in Figure 3.1. The soil strength parameters are also listed in Figure 3.1.

Figure 3.1 A Sample Slope Stability Problem



Defining the Problem

The SLOPE/W DEFINE function is used to define a problem.

➢ **To start DEFINE:**

- Select DEFINE from the Start Programs menu under SLOPE/W.

  -- or --

- Double-click the DEFINE icon in the SLOPE/W Group window.

  When the DEFINE window appears, click the Maximize button in the upper-right corner of the DEFINE window so that the DEFINE window will cover the entire screen. This maximizes the workspace for defining the problem.

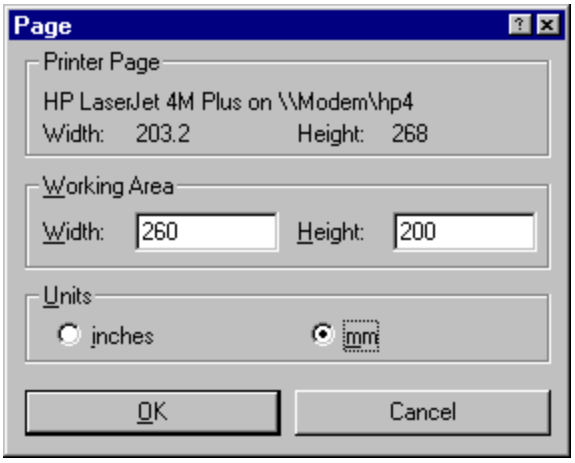
NOTE: It is assumed that you are readily familiar with the fundamentals of the Windows environment. If you are not, then you will first need to learn how to navigate within the Windows environment before learning how to use SLOPE/W. The SLOPE/W User's Guide does not provide instructions on the fundamentals of using Windows. You will have to get this information from other documentation.

Set the Working Area

The working area is the size of the space available for defining the problem. The working area may be smaller, equal to or greater than the printer page. If the working area is larger than the printer page, the problem will be printed on multiple pages when the Zoom Factor is 1.0 or greater. The working area should be set so that you can work at a convenient scale. For this example, a suitable working area is 260 mm wide and 200 mm high.

➢ **To set the working page size:**

1. Choose Page from the Set menu. The Set Page dialog box appears:



   The Printer Page group box displays the name of the printer selected and the printing space available on one printer page. This information is presented to help you define a working area that will print properly.

2. Select mm in the Page Units group box.

3. Type 260 in the Working Area Width edit box. Press the TAB key to move to the next edit box.

4. Type 200 in the Height edit box.

5. Select OK.

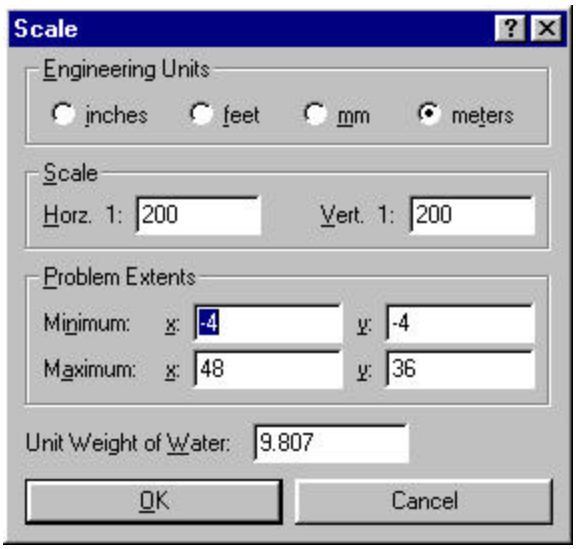
Set the Scale

The geometry of the problem is defined in meters. A suitable scale is 1:200. This makes the drawing small enough to fit within the page margins.

The geometry of the problem is defined in meters. As shown in Figure 3.1, the problem is 14 m high and about 40 m wide. The lower-left corner of the problem will be drawn at (0,0). The extents need to be larger than the size of the problem to allow for a margin around the drawing. Let us initially estimate the extents to be from -4 to 40 m in both directions. Once the extents of the problem have been set, DEFINE computes an approximate scale. The scale can then be adjusted to an even value. The maximum *x* and *y* extents will then be automatically adjusted to reflect the scale you have selected..

➢ **To set the scale:**

1. Choose Set Scale from the DEFINE menu. The Set Scale dialog box appears:



2. Select Meters in the Engineering Units group box.

3. Type the following values in the Problem Extents edit boxes:

Minimum:	x: **-4**	Minimum:	y: **-4**
Maximum:	x: **40**	Maximum:	y: **40**

The Horz. 1: scale will change to 169.23 and the Vert. 1: scale to 220. We do not want to work at such an odd scale. An even scale of 1:80 in both directions appears acceptable for this problem.

4. Type **200** in the Horz. 1: edit box, and type **200** in the Vert. 1: edit box.

The *Maximum x* will change to 48 and the *Maximum y* will change to 36. This means that at a scale of 1:200, the allowable problem extents are from -4 to 48 m in the *x*-direction and from -4 to 36 m in the *y*-direction for the previously selected working area 260 mm wide and 200 mm high.

5. Select OK.

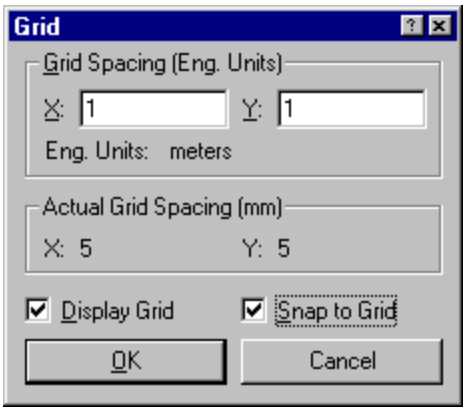
Since the problem is defined in terms of meters and kN, the unit weight of water must be 9.807 kN/m^3, which is the default value when the engineering dimensions are defined in meters.

Set the Grid Spacing

A background grid of points is required to help you draw the problem. These points can be "snapped to" when creating the problem geometry in order to create points and lines with exact coordinates. A suitable grid spacing in this example is 1 meter.

➢ **To set and display the grid:**

1. Choose Grid from the Set menu. The Set Grid dialog box appears:



2. Type **1** in the Grid Spacing X: edit box.

3. Type **1** in the Y: edit box.

   The actual grid spacing on the screen will be a distance of 5.0 mm between each grid point. This value is displayed in the Actual Grid Spacing group box.

4. Check the Display Grid check box.

5. Check the Snap to Grid check box.

6. Select OK.

   The grid is displayed in the DEFINE window. As you move the cursor in the window, the coordinates of the nearest grid point (in engineering units) are displayed in the status bar.

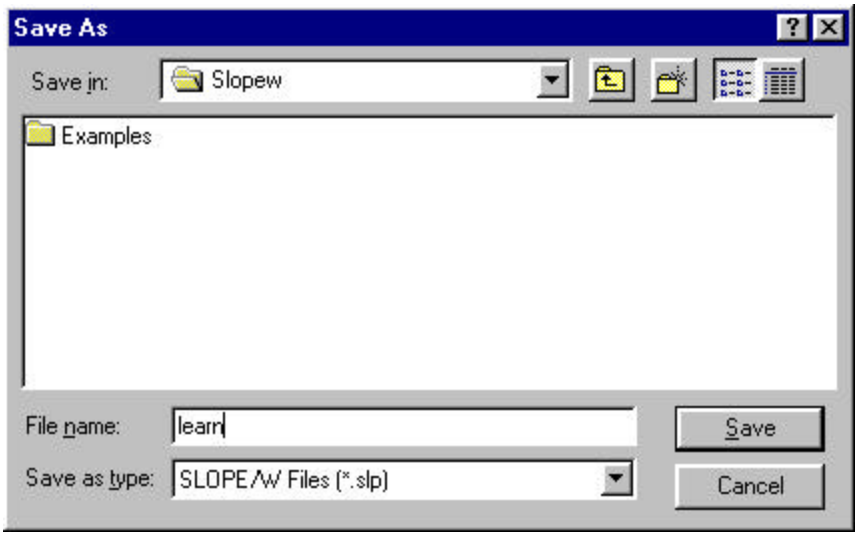
Saving the Problem

The problem definition data must be saved in a file. This allows the SOLVE and CONTOUR functions to obtain the problem definition for solving the problem and viewing the results.

The data may be saved at any time during a problem definition session. It is good practice to save the data frequently.

➢ **To save the data to a file:**

1. Choose Save from the File menu. The following dialog box appears:



2. Type a file name in the File Name edit box. For example, type **LEARN**.

3. Select Save. The data will be saved to the file LEARN.SLP. Once it is saved, the file name is displayed in the DEFINE window title bar.

   The file name may include a drive name and directory path. If you do not include a path, the file will be saved in the directory name displayed in the Save In box..

   The file name extension must be **SLP**. SLOPE/W will add the extension to the file name if it is not specified.

The next time you choose File Save, the file will be saved without first bringing up the Save File As dialog box. This is because a file name is already specified.

It is often useful when modifying a file to save it under a different name. This preserves the previous contents of the file.

➢ **To save data to a file with a different name:**

1. Choose File Save As. The same dialog box appears.

2. Type the new file name.

   If the file name you type already exists, you will be asked whether you wish to replace the file which already exists. If you select No, you must retype the file name. If you select Yes, the previous copy of the file will be lost.

Sketch the Problem

In defining a slope stability problem, it is convenient to first prepare a sketch of the problem dimensions. This sketch is a useful guide for drawing the geometric elements of the problem.

➢ **To sketch the slope stability problem:**

1. In the Zoom toolbar, click on the Zoom Page button with the left mouse button.

   The entire working area is displayed in the DEFINE window.

2. Choose Lines from the Sketch menu. The cursor will change from an arrow to a cross-hair, and the status bar will indicate that "Sketch Lines" is the current operating mode.

3. Using the mouse, move the cursor near position (0,14), as indicated in the status bar at the bottom of the window, and click the left mouse button. The cursor snaps to the grid point at (0,14). As you move the mouse, a line is drawn from (0,14) to the new cursor position.

   The cursor position (in engineering units) is always displayed in the status bar. It is updated as you move the cursor with the mouse.

4. Move the cursor near (10,14) and click the left mouse button. The cursor snaps to (10,14) and a line is drawn from (0,14) to (10,14).

5. Move the cursor near (30,4) and click the left mouse button. A line is drawn from (10,14) to (30,4).

6. Move the cursor near (40,4) and click the left mouse button. A line is drawn from (30,4) to (40,4).

7. Move the cursor near (40,0) and click the left mouse button. A line is drawn from (40,4) to (40,0).

8. Move the cursor near (0,0) and click the left mouse button. A line is drawn from (40,0) to (0,0).

9. Move the cursor near (0,14) and click the left mouse button. A line is drawn from (0,0) to (0,14).

10. Click the right mouse button to finish sketching a line. The cursor will change from a cross-hair back to an arrow; you are then back in Work Mode.

11. Choose Lines from the Sketch menu again.

12. Move the cursor near (0,9) and click the left mouse button. The cursor snaps to (0,9).

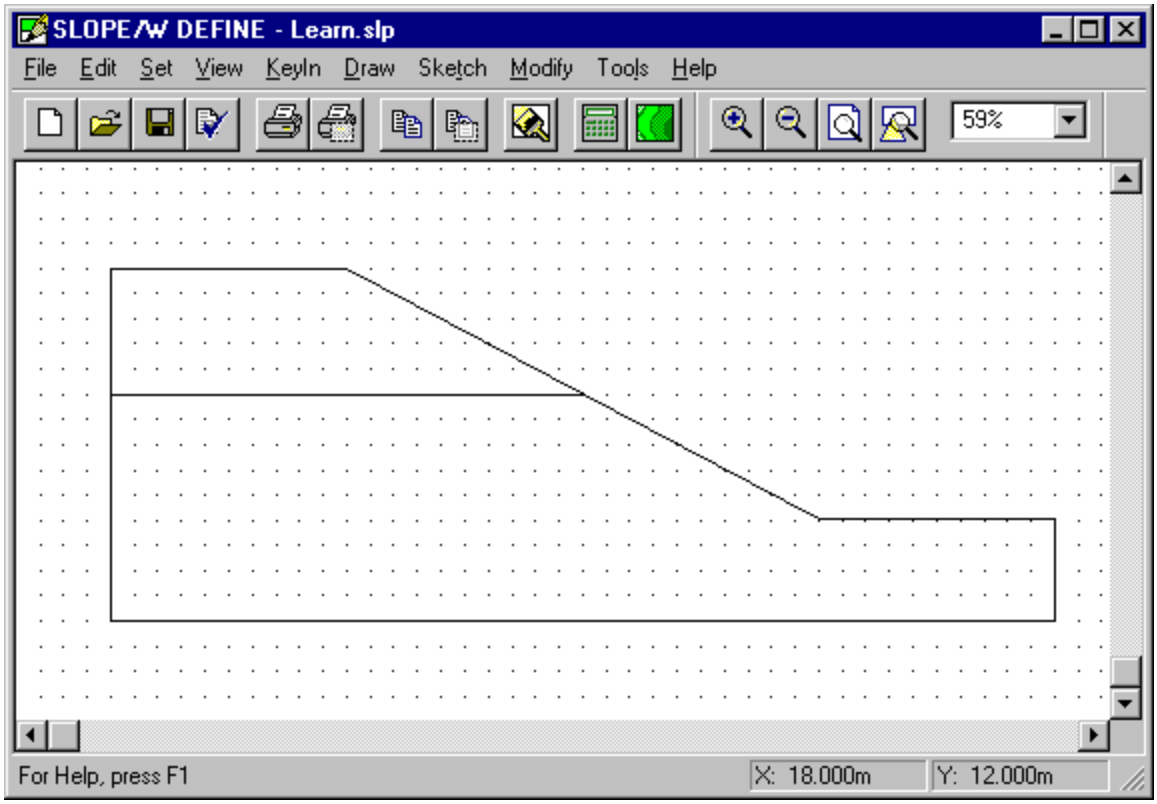
13. Move the cursor near (20,9) and click the left mouse button. A line is drawn from (0,9) to (20,9), which is the boundary between the upper and lower soil layers.

14. Click the right mouse button to finish sketching a line. The cursor will change from a cross-hair back to an arrow; you are then back in Work Mode.

15. In the Zoom Toolbar, click on the Zoom Objects button with the left mouse button.

    The drawing is enlarged so that the lines you just sketched fill the DEFINE window.

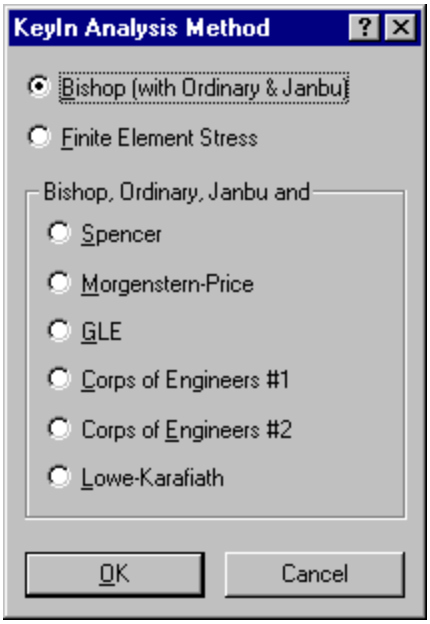
After you have completed the above steps, your screen should look like the following:



Specify the Analysis Methods

➢ **To specify the analysis methods:**

1. Choose Analysis Method from the KeyIn menu. The following dialog box will appear:



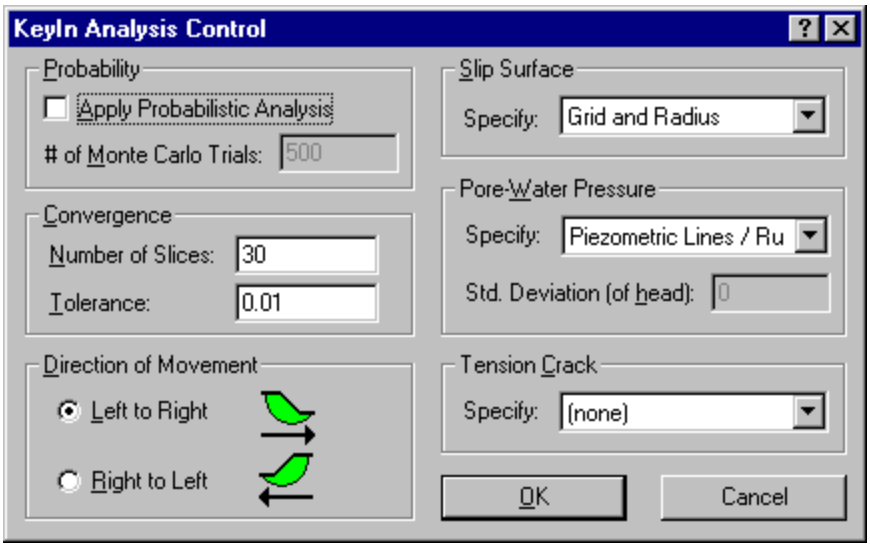
2. Select the Bishop (with Ordinary & Janbu) option (this is the default option).

3. Select OK.

Specify the Analysis Options

➢ **To specify the options used in the analysis:**

1. Choose Analysis Control from the KeyIn menu. The following dialog box appears:



2. Use the options selected by default in the KeyIn Analysis Control dialog box:

   - Probabilistic analysis will not be applied.
   - The default convergence information will be used.
   - The direction of the slip surface movement will be from left to right.
   - Grid and Radius is the selected Slip Surface option. This allows you to specify slip surfaces by defining a grid of slip surface centers and radius lines.
   - Piezometric Lines / Ru is the selected Pore-Water Pressure option.
   - No tension crack will be specified.

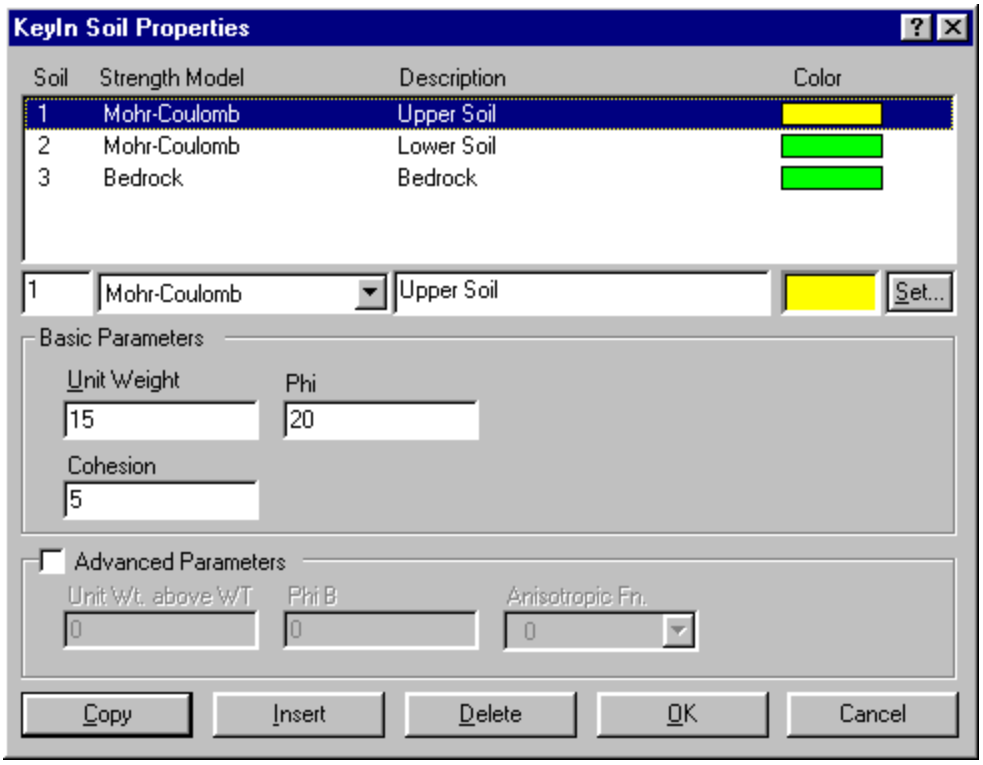
3. Select OK.

Define Soil Properties

The soil properties of this problem are listed in Figure 3.1. The properties must be defined for three materials.

➢ **To define the soil properties:**

1. Choose Soil Properties from the KeyIn menu. The KeyIn Soil Properties dialog box appears:



2. Type **1** in the Soil edit box (underneath the list box) to indicate that you are defining Soil 1.

3. Press TAB twice to move to the Description edit box (The Strength Model does not need to be selected, since it is the default Mohr-Coulomb model).

4. Type **Upper Soil Layer** in the Description edit box.

5. Type **15** in the Unit Weight edit box.

6. Type **5** in the Cohesion edit box.

7. Type **20** in the Phi edit box.

8. Select Copy. The values contained in the edit boxes are copied into the list box.

9. Repeat Steps 2 to 8 for Soil 2 , using **Lower Soil Layer** for the description, **18** for the Unit Weight, **10** for Cohesion, and **25** for Phi.

10. Type **3** in the Soil edit box.

11. Click on down arrow to the left of the Strength Model edit box and select the Bedrock strength model. The Soil Description is set to Bedrock and the Unit Weight changes to **-1**.

12. Select Copy to copy the bedrock properties into the list box. The list box should now look the same as the dialog box shown above.

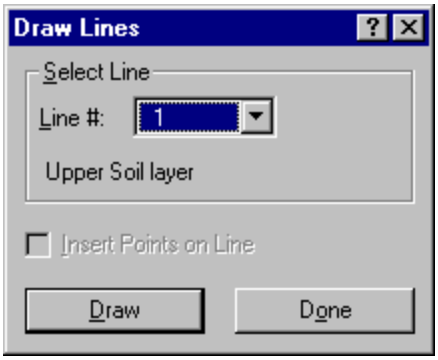
13. Select OK.

Draw Lines

The geometry and stratigraphy are defined by lines connected to points. A line must be defined for each soil layer. All lines must begin at the left-most point and end at the right-most point. The normal procedure is to define the top line first (Soil 1) and then the remaining lines in sequential order.

➢ **To draw the lines in the geometry:**

1. Choose Lines from the Draw menu. The following dialog box appears:



2. Select 1 in the Line # drop-down list box to draw Line 1 (this is the default value).

3. Select the Draw button. The cursor will change from an arrow to a cross-hair, and the status bar will indicate that "Draw Lines" is the current operating mode.

4. Move the cursor near (0,14) and click the left mouse button (The coordinates (0,14) should be displayed in the status bar before you click). The cursor snaps to the grid point at (0,14) and creates a point there. As you move the cursor, a line is drawn from the point (Point 1) to the new cursor position.

5. Move the cursor to the crest of the slope (10,14) and click the left mouse button. The cursor snaps to the grid point at (10,14), a point is created (Point 2), and a red line is drawn from Point 1 to Point 2.

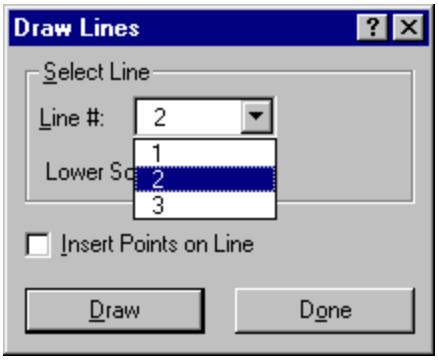
6. Move the cursor along the slope to where there is a break between the soil types (20,9) and click the left mouse button. The cursor snaps to the grid point at (20,9), a point is created (Point 3), and a red line is drawn from Point 2 to Point 3.

7. Move the cursor near the toe of the slope (30,4) and click the left mouse button.

8. Move the cursor to the right side of the problem near (40,4) and click the left mouse button. Then click the right mouse button (or press the ESC key) to finish drawing Line 1.

   The Draw Lines dialog box appears again.

9. Click the down arrow to the right of the Line # edit box. A list of available lines (one for each soil number defined) appears:



10. Click on 2 in the drop-down list box and then select the Draw button to start drawing Line 2. The cursor will change from an arrow to a cross-hair, and the status bar will indicate that "Draw Lines" is the current operating mode.

11. Move the cursor to the left side of the problem near the contact between the upper and lower soil layers (0,9) and click the left mouse button.

12. Click the left mouse button near Point 3 (20,9). (The cursor snaps to Point 3 instead of creating a new point at (20,9), since Point 3 already exists at the grid point). Then click the right mouse button to finish drawing Line 2.

    Since the Line 2 endpoint (Point 3) lies in the middle of the previous line (Line 1), SLOPE/W generates the remainder of Line 2 along Line 1 from Point 3 to Point 5. The complete Line 2 appears as a red line, and the Draw Lines dialog box reappears.

13. Click the down arrow to the right of the Line # edit box and click on 3.

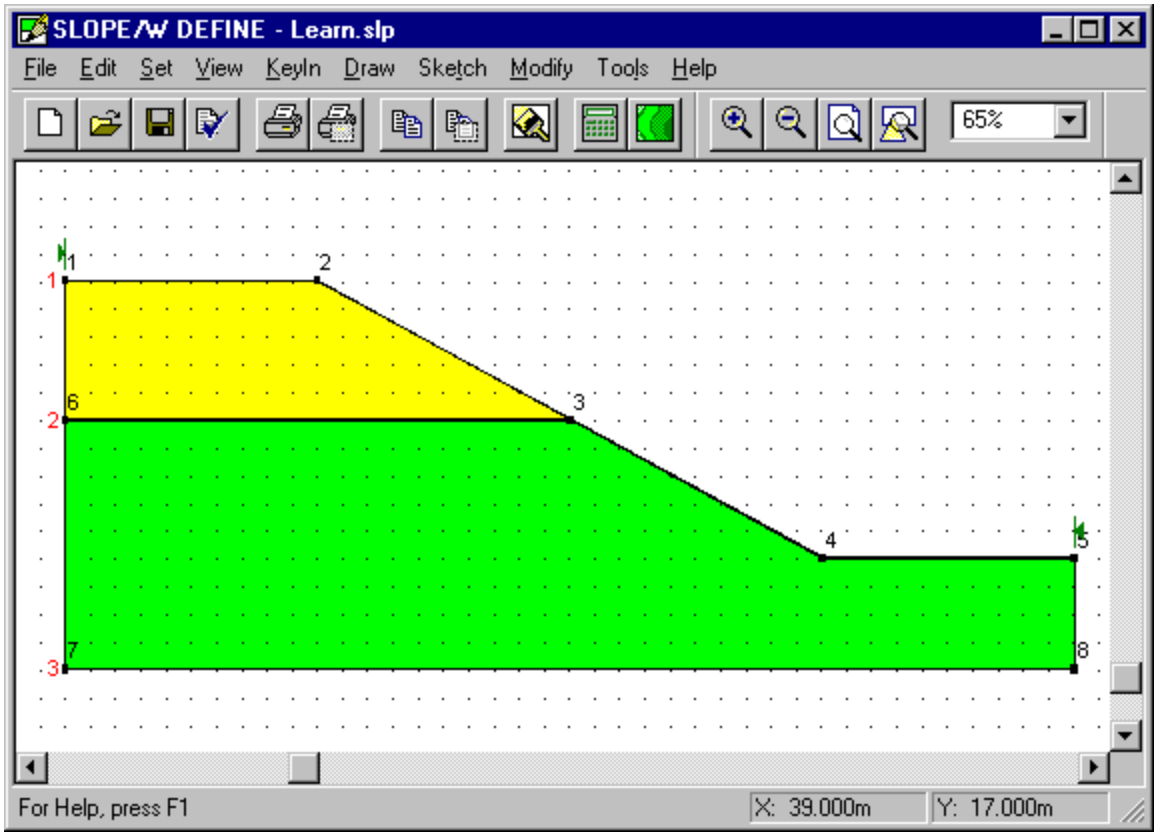
14. Select OK to start drawing Line 3. Soil 1 will be shaded yellow. The cursor will change from an arrow to a cross-hair, and the status bar will indicate that "Draw Lines" is the current operating mode.

15. Move the cursor to the lower-left corner near the contact between the lower soil layer and the bedrock (0,0) and click the left mouse button.

16. Move the cursor to the lower-right corner near the contact between the lower soil layer and the bedrock (40,0) and click the left mouse button. Then click the right mouse button to finish drawing Line 3.

17. Select Done in the Draw Lines dialog box to finish drawing lines. Soil 2 will be shaded light green.

After you have completed the above steps, your screen should look like the following:



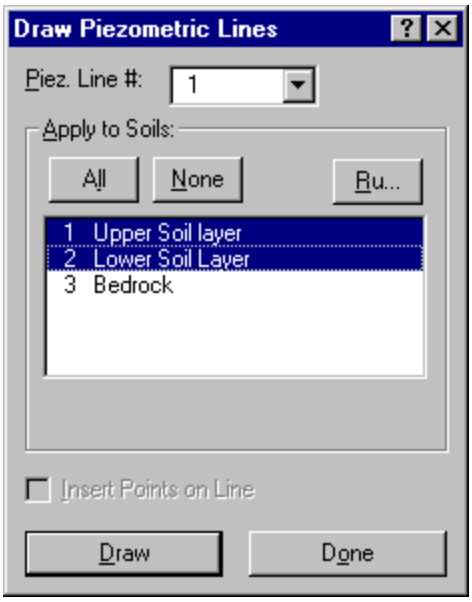
Draw Piezometric Lines

The pore-water pressure conditions in both Soil 1 and Soil 2 are defined by one piezometric line.

➢ **To draw the piezometric line:**

1. If you have turned off the grid, choose the Snap Grid command from the Grid Toolbar.

2. Choose Pore Water Pressure from the Draw menu. The following dialog box appears:



3. Select 1 in the Piez. Line # drop-down list box to draw one piezometric line (this is the default value).

4. Select Soil 1 (Upper Soil Layer) and Soil 2 (Lower Soil Layer) in the Apply To Soils list box to apply the piezometric line to Soils 1 and 2.

5. Select the Draw button. The cursor will change from an arrow to a cross-hair, and the status bar will indicate that "Draw P.W.P." is the current operating mode.

6. Move the cursor near (0,11) (at the left of the problem) and click the left mouse button. The cursor snaps to the grid point at (0,11) and a point is created (Point 9). As you move the cursor, a dashed line is drawn from Point 9 to the new cursor position.

7. Move the cursor near (15,8) and click the left mouse button. The cursor snaps to the grid point at (15,8), a point is created (Point 10), and a red dashed line is drawn from Point 9 to Point 10.

8. Move the cursor near (30,3) and click the left mouse button.

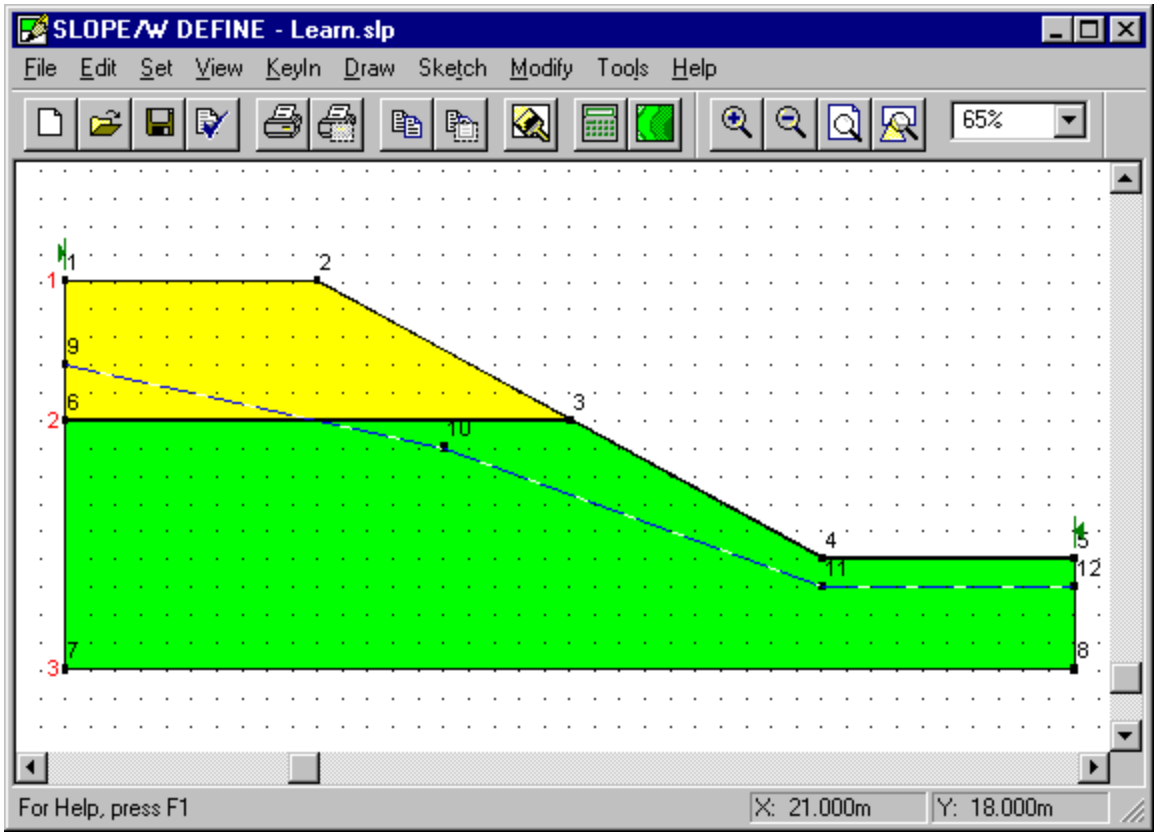
9. Move the cursor near (40,3) and click the left mouse button. Then click the right mouse button to finish drawing the piezometric line for Soils 1 and 2.

   The Draw Piez. Lines dialog box appears again.

10. Select Done in the Draw Piez. Lines dialog box to finish drawing piezometric lines.

    Since the slip surfaces do not extend into the bedrock, it is not necessary to define a piezometric line for the bedrock.

After you have completed the above steps, your screen should look like the following:



Draw the Slip Surface Radius

To control the location of the trial slip surfaces, it is necessary to define lines or points which are used to compute the slip circle radii.

➢ **To draw the radius lines**

1. If you have turned off the background grid, click on the Snap to Grid button in the Grid toolbar.

2. Choose Slip Surface from the Draw menu. The Slip Surface cascading menu will appear.

   Select Radius from the Slip Surface cascading menu. The cursor will change from an arrow to a cross-hair, and the status bar will indicate that "Draw Slip Surface Radius" is the current operating mode.

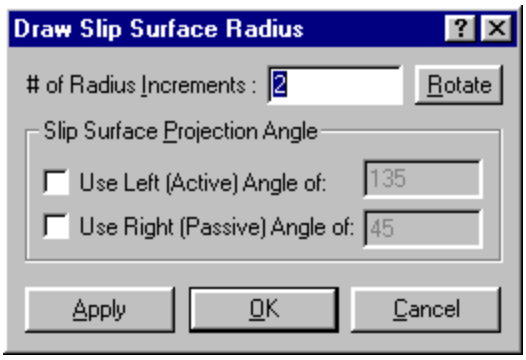
3. Move the cursor near (15,4) and click the left mouse button. The cursor snaps to the grid point at (15,4) and a point is created (Point 13). As you move the cursor, a line is drawn from Point 13 to the new cursor position.

4. Move the cursor near (15,2) and click the left mouse button. The cursor snaps to the grid point at (15,2), a point is created (Point 14), and a red line is drawn from Point 13 to Point 14.

5. Move the cursor near (29,2) and click the left mouse button.

6. Move the cursor near (29,4) and click the left mouse button.

   The region in which the radius lines will be drawn is now outlined. The Draw Slip Surface Radius dialog window appears:

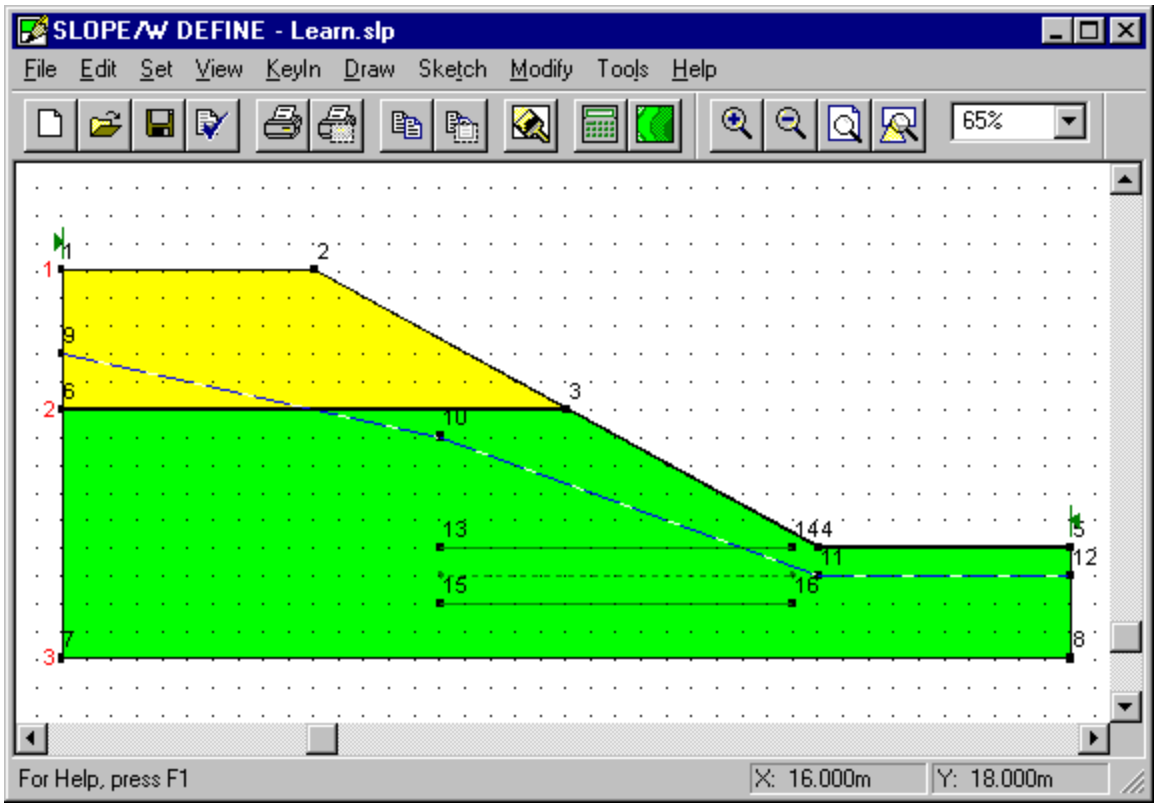


7. Accept the default value of 2 for the #of Radius Increments.

9. Select OK to generate the radius lines.

   Three radius lines are displayed in the DEFINE window. SLOPE/W SOLVE will define slip circles that are tangent to these lines.

After you have completed the above steps, your screen should look like the following:



Draw the Slip Surface Grid

A grid of rotation centers must be defined to specify and control the location of trial slip surfaces.

➢ **To draw the grid of centers:**

1. If you have turned off the background grid, click on the Snap to Grid button in the Grid toolbar.

2. Choose Slip Surface from the Draw menu. The Slip Surface cascading menu will appear.

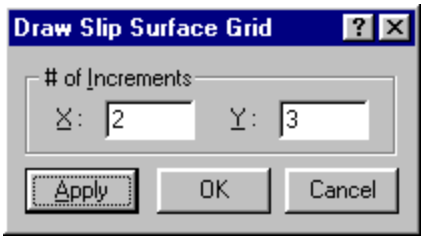
   Select Grid from the Slip Surface cascading menu. The cursor will change from an arrow to a cross-hair, and the status bar will indicate that "Draw Slip Surface Grid" is the current operating mode.

4. Move the cursor near (23,25) and click the left mouse button. (You may need to scroll the window first to get to this position). The cursor snaps to the grid point at (23,25) and a point is created (Point 17). As you move the cursor, a line is drawn from Point 17 to the new cursor position.

5. Move the cursor near (22,19) and click the left mouse button. The cursor snaps to the grid point at (22,19) and a point is created (Point 18). As you move the cursor, a parallelogram is drawn from Point 17 to Point 18 to the new cursor position.

6. Move the cursor near (26,19) and click the left mouse button. A parallelogram is drawn from Point 17 to Point 18 to Point 19.

   The region in which the grid centers will be drawn is now outlined. The Draw Slip Surface Grid dialog window appears:



   The value in the increment box represents the number of horizontal divisions and vertical divisions in which to divide the grid region.

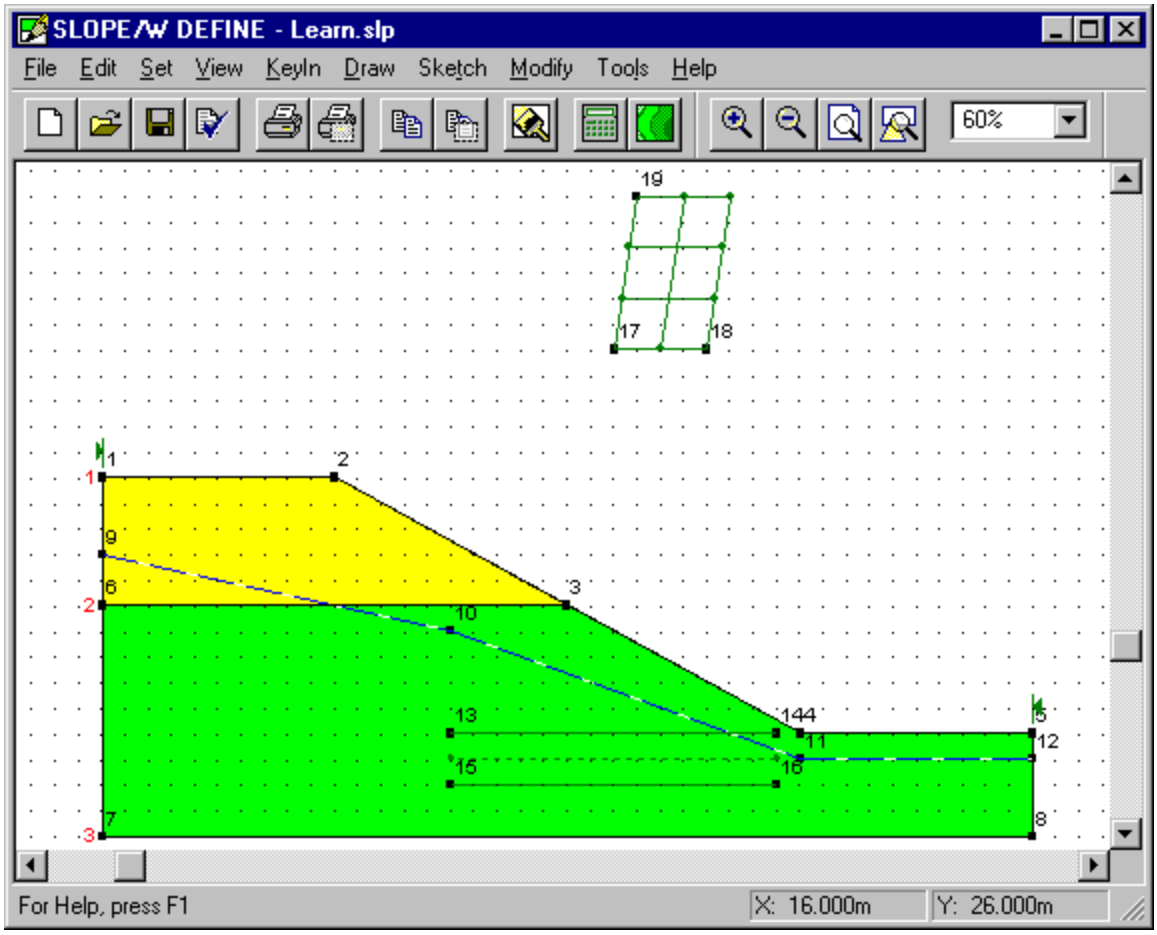
7. Type **2** in the X increment edit box.

8. Type **3** in the Y increment edit box

9. Select OK or Apply to generate the grid centers.

   A grid of 12 center points is displayed in the DEFINE window. SLOPE/W SOLVE will define slip circles using these center points.

After you have completed the above steps, your screen should look like the following:

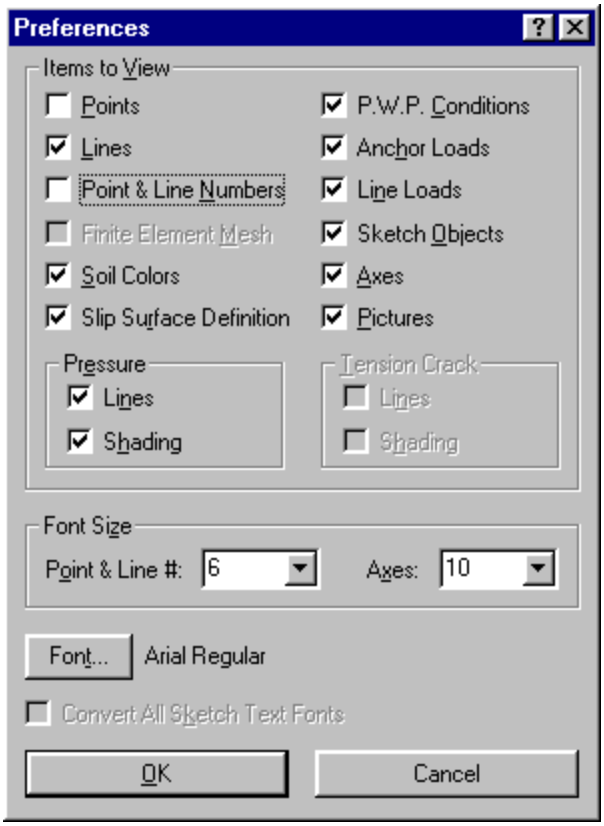


View Preferences

You no longer need to view the points or the point numbers in the DEFINE window.

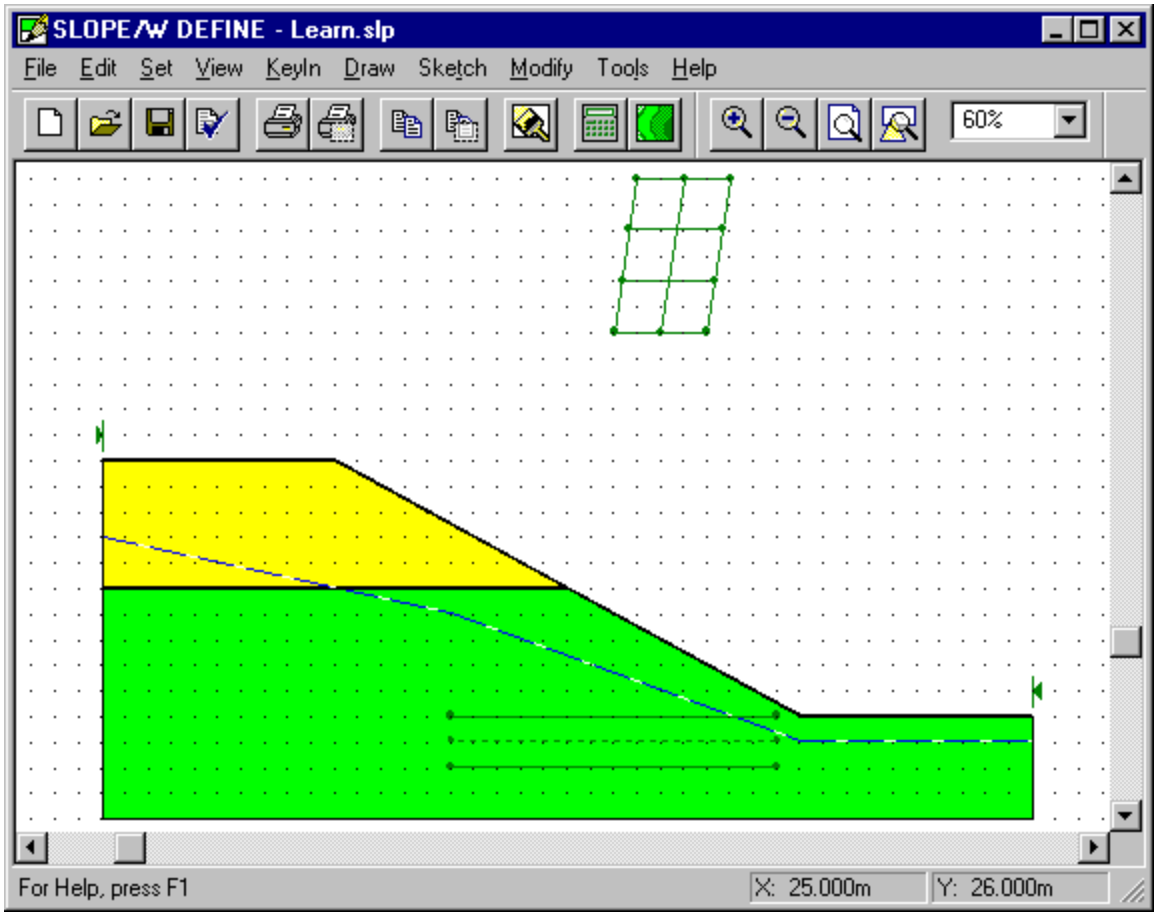
➢ **To turn off the points and the point numbers:**

1. Choose Preferences from the View menu. The following dialog box appears:



3. Uncheck the Points check box in order to not display any points on the drawing.
4. Uncheck the Point & Line Numbers check box in order to not display any point or line numbers on the drawing.
5. Select OK.

The problem will be drawn without the points or point and line numbers displayed.



NOTE: You can also select and unselect the View Preferences by clicking on the icons in the View Preferences toolbar. You can learn about each of the icons by placing the cursor over the icon. A tool tip will appear for a few seconds and a description is displayed on the status bar at the bottom of the window.

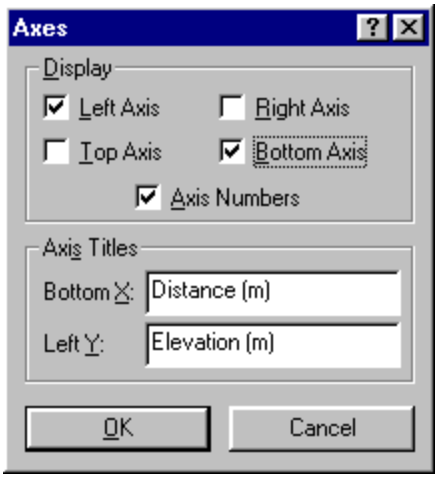
Sketch Axes

Sketching an axis on the drawing facilitates viewing the drawing and interpreting the drawing after it is printed.

➢ **To sketch an axis:**

1. If you have turned off the background grid, click on the Snap to Grid button in the Grid toolbar. This allows you to define an evenly-spaced region for the axis.

2. Choose Axes from the Sketch menu. The following dialog box appears:



3. Check the Left Axis, Bottom Axis, and Axis Numbers check boxes in the Display group box. The Top Axis and Right Axis check boxes should be unchecked.

   This will cause an X axis to be sketched along the bottom side of the specified region and a Y axis to be sketched along the left side of the specified region.

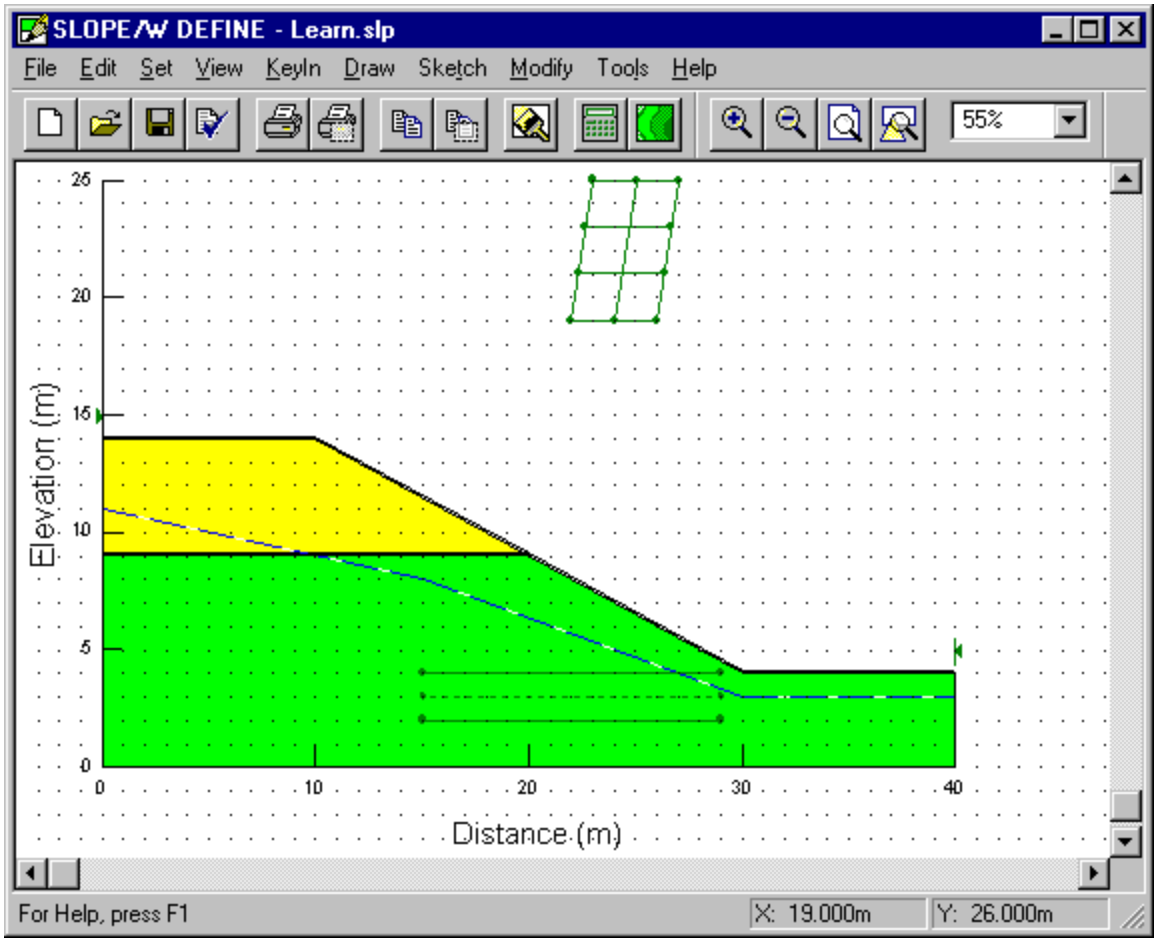
4. Select OK. The cursor will change from an arrow to a cross-hair, and the status bar will indicate that “Sketch Axes” is the current operating mode.

5. Move the cursor near position (0,0). Hold the left mouse button down, but do not release it. As you move the mouse, a rectangle appears.

6. "Drag" the mouse near (40,25), and release the left mouse button.

   An *x*- and *y*-axis is generated within the region.

After you have completed the above steps, your screen should look like the following:



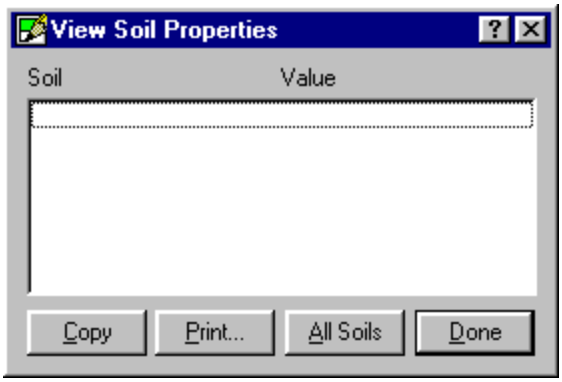
If you wish to modify the axis increments, choose Axes from the Set menu. See the DEFINE Reference section for more information on the Set Axes command.

Display Soil Properties

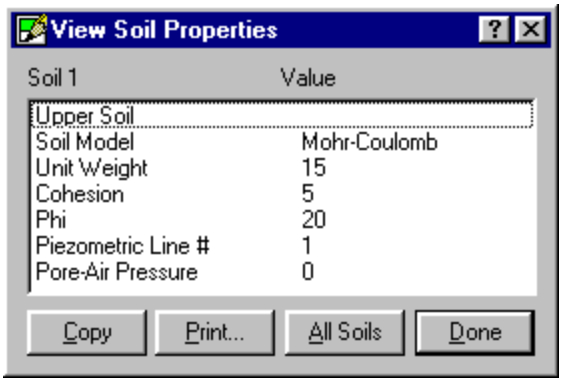
Now that the problem definition has been completed, you can quickly double-check the soil properties to ensure they are defined correctly. The View Soil Properties command allows you to graphically select a soil line or region and view its properties; you can also display a list of all soil properties and print or copy the list to the Windows clipboard for importing into other applications.

➢ **To view the soil properties:**

1. Choose Soil Properties from the View menu. The cursor will change from an arrow to a cross-hair, and the status bar will indicate that "View Soil Properties" is the current operating mode. The following dialog box is displayed:



2. Move the cursor near (5,11) (or anywhere inside Soil 1 or on top of Soil Line 1) and click the left mouse button. The soil is selected with a diagonal hatch pattern, and the soil line and points are highlighted. The soil properties of Soil 1 are displayed in the dialog box as follows:



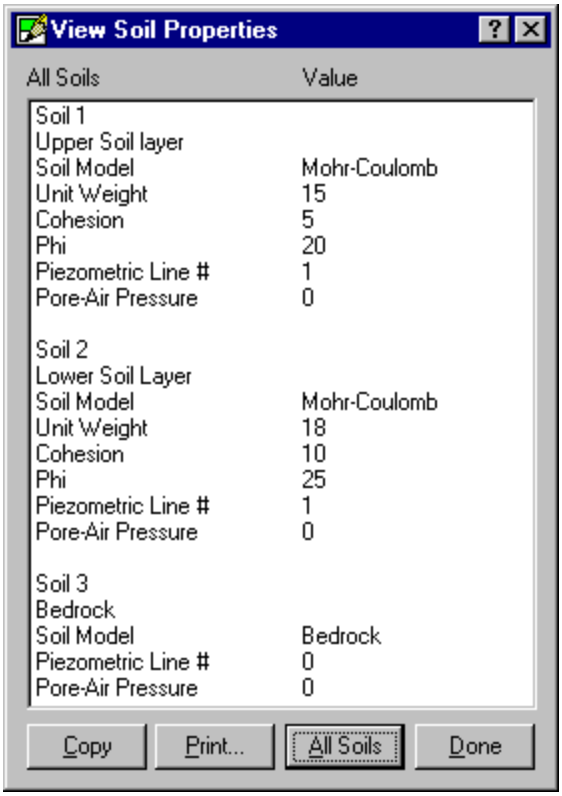
The dialog box lists the soil number, description, model, the properties specific to the soil model, any piezometric line or r_u value defined for the soil, and the pore-air pressure.

3. To see all the soil properties, re-size the dialog box by dragging the bottom edge of the window down until all information is displayed.

4. To view the properties for Soil 2, click the left mouse button near (5,5) (or anywhere inside Soil 2 or on top of Soil Line 2) and click the left mouse button. The soil is selected with a diagonal hatch pattern, and the soil line and points are highlighted. The soil properties of Soil 2 are displayed in the dialog box.

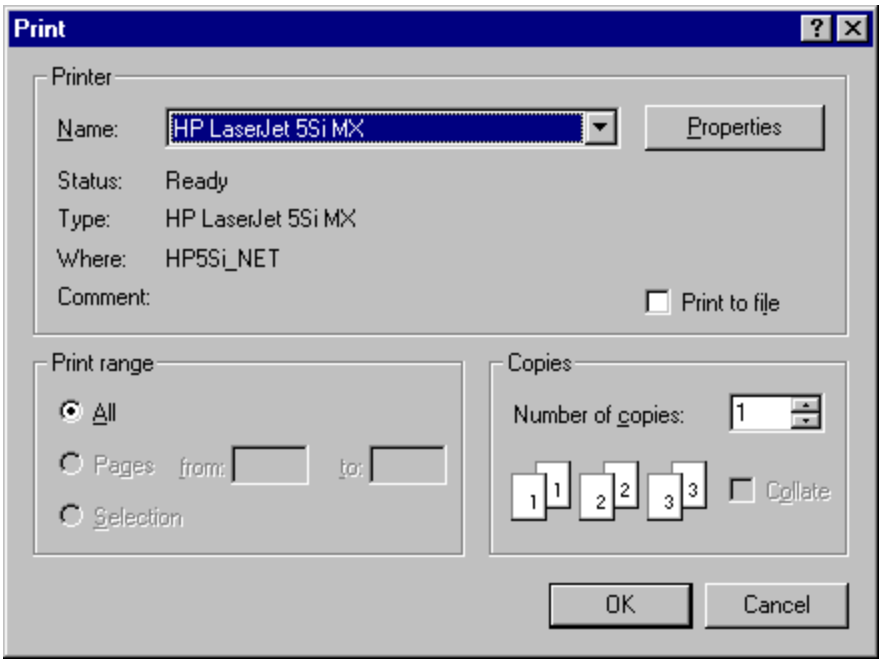
5. To view a list of all soil properties in the dialog box, select the All Soils button.

   The currently-selected soil is unselected, and all soil properties are displayed in the dialog box as follows:



6. To copy all of the soil properties to the Windows Clipboard, select Copy. The soil properties are copied to the Clipboard and can now be pasted into another Windows application.

7. To print all of the soil properties on the current printer, select Print. The following dialog box appears:



8. Select a printer from the Printer Name drop-down list box and then select OK to print the soil properties on this printer.

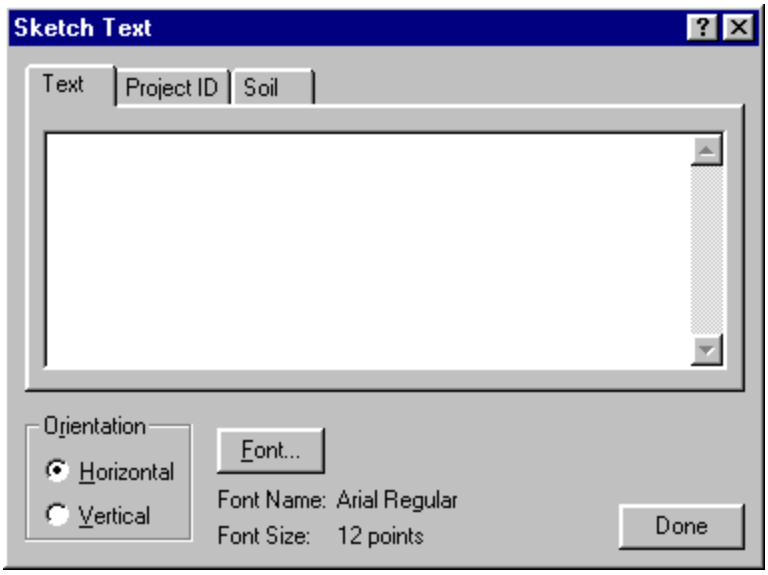
9. Select the Done button or click the right mouse button to finish viewing soil properties.

Label the Soils

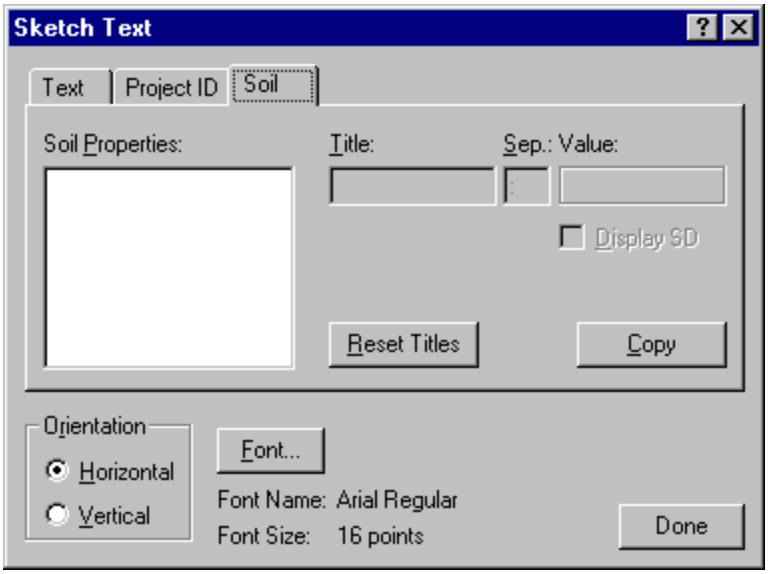
Not only can you view the soil properties interactively, but you can also place the soil properties on the drawing as a sketch text label. This allows you to print the soil properties on the drawing for reference purposes. For this example, we will simply add text labels that will identify each soil name.

➢ **To add soil labels:**

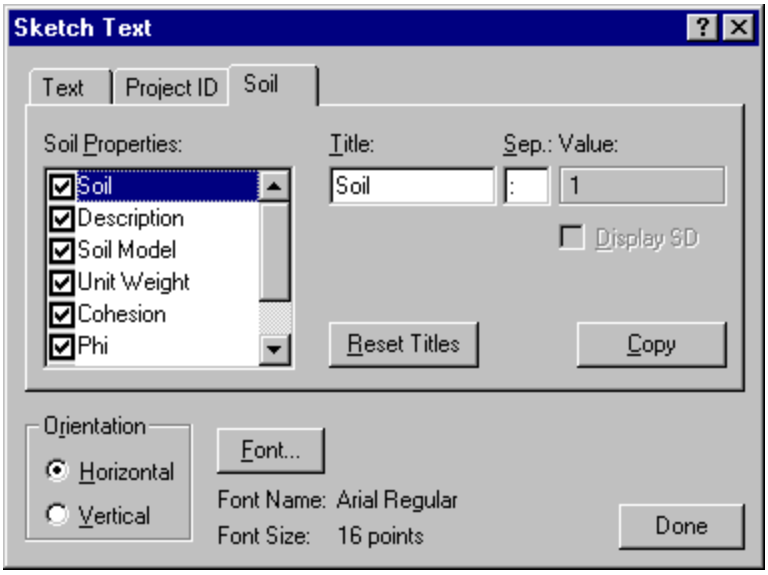
1. Choose Text from the Sketch menu. The following dialog box appears:



2. Select the Soil tab at the top of the dialog box. A soil information property sheet is displayed in the dialog box:



3. In the SLOPE/W window, move the cursor inside the top soil layer. (Notice that the cursor changes to a black selection arrow when it is inside a soil layer.) Click the left mouse button near position (2,11) to select Soil 1. The soil is shaded with a diagonal hatch pattern, and the soil line and points are highlighted. The Soil 1 properties are displayed in the Sketch Text dialog box:

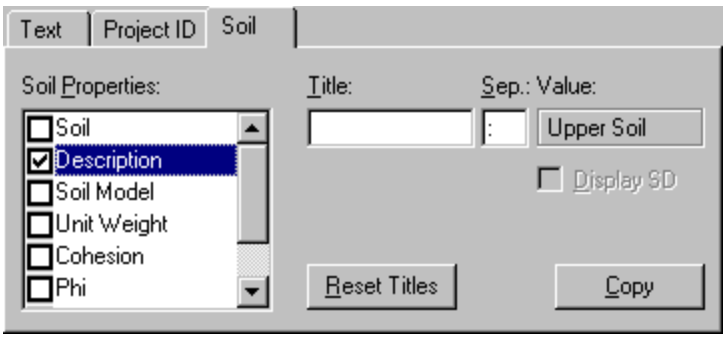


By default, all soil parameters are checked in the Soil Properties list box.

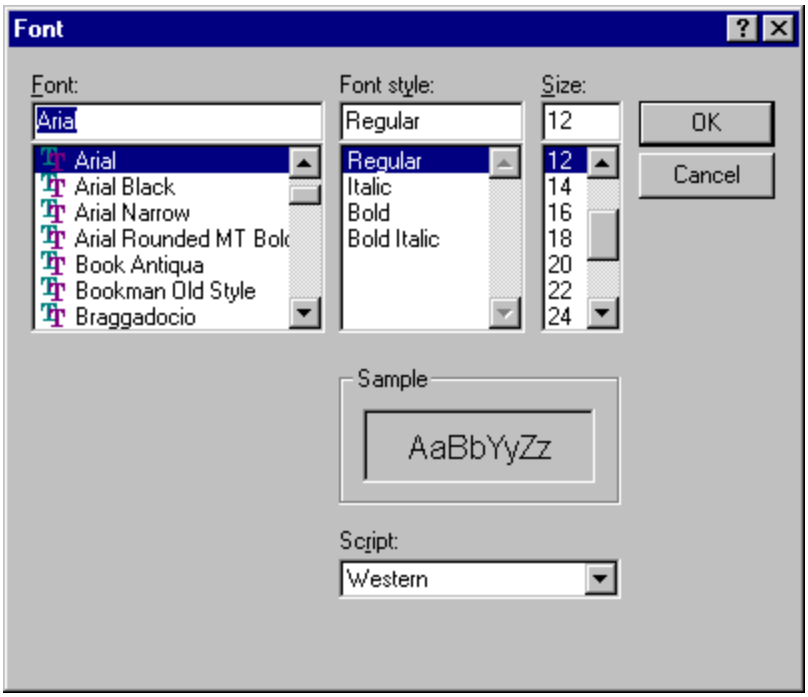
4. Since we only want to label the soil with its description, uncheck every parameter in the list box except Description. You will have to use the scroll bar to see all of the parameters in the list box.

5. Select Description in the Soil Properties list box, and "Description" appears in the Title edit box. Double-click the left mouse button inside the Title edit box and press the Delete key to remove the Description title text.

   When you have completed the previous two steps, the Soil property sheet should appear as follows (note that only the Description parameter is checked and it has no Title):



6. Click on the Font button to select the font to use for the soil label. The following dialog box appears:



7. Select the desired font (e.g., Arial) in the Font list box and style in the Font Style list box.

8. Select a point size (e.g., 12) from the Size list box or type the desired point size in the Size edit box.

9 Select OK to return to the Sketch Text dialog box.

10. Move the cursor inside Soil 1(the selected soil layer), so that the cursor is shown as a cross-hair. Then, click the left mouse button near position (2,11) to place the soil label.

NOTE: When you move the cursor inside a soil layer that isn't already selected, the cursor changes to a black selection arrow. This indicates that a label will not be placed if you click the left mouse button; instead, a new soil will be selected.

The label **Upper Soil Layer** appears on the drawing above and to the right of the selected position.

11. To place a soil label on Soil 2, move the cursor inside the bottom soil layer. (Notice that the cursor changes to a black selection arrow.) Then, click the left mouse button near position (2,4) to select Soil 2. The soil is shaded with a diagonal hatch pattern, and the soil line and points are highlighted. The Soil 2 properties are displayed in the Sketch Text dialog box.

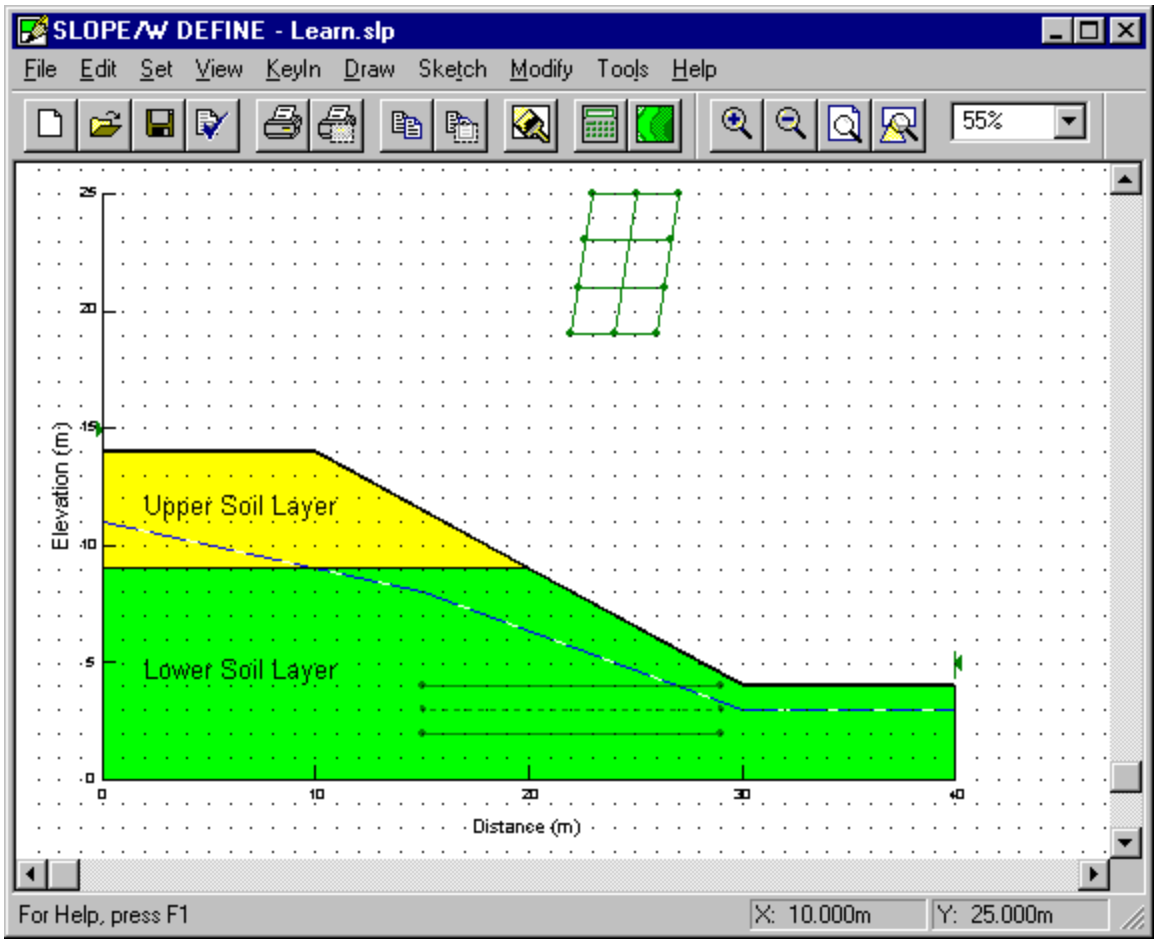
12. Click the left mouse button inside Soil 2 near position (2,4) to place the soil label.

The label **Lower Soil Layer** appears on the drawing above and to the right of the selected position.

NOTE: Notice that the soil label for Soil 2 is different than the label for Soil 1. This is because when you placed the soil label, the soil description was obtained from the Soil Properties information. If you change the soil descriptions using KeyIn Soil Properties, the soil labels will be automatically updated to show the new descriptions. If you wish to display more of the soil properties on your soil label, choose the Modify Text command and click on the soil label.

13. To finish placing soil labels, press the Done button in the Sketch Text dialog box. You can also click the right mouse button or press the ESC key to exit from the Sketch Text dialog box.

After you have completed the above steps, your screen should look like the following:

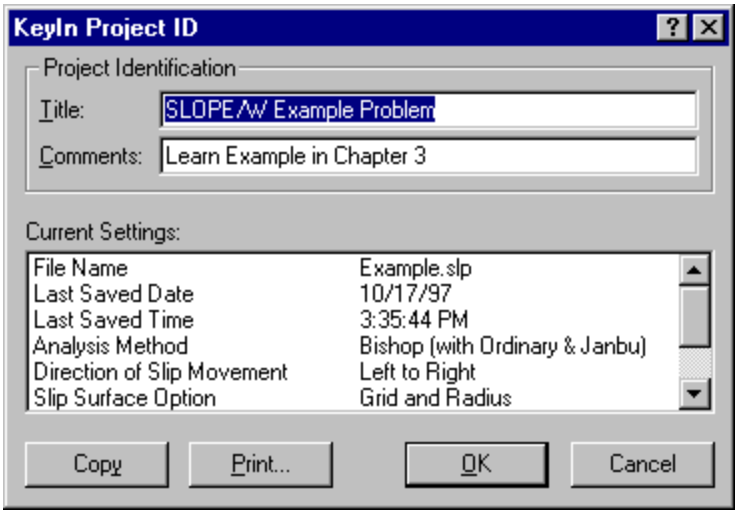


Add a Problem Identification Label

You can now place a Project ID text label on your drawing that will help to identify it when you later view or print the drawing. The procedure for adding a Project ID text label is similar to adding a Soil Properties text label. First, however, you need to enter the Project ID information.

➢ **To specify the Project ID information:**

1. Choose Project ID from the KeyIn menu. The following dialog box appears:



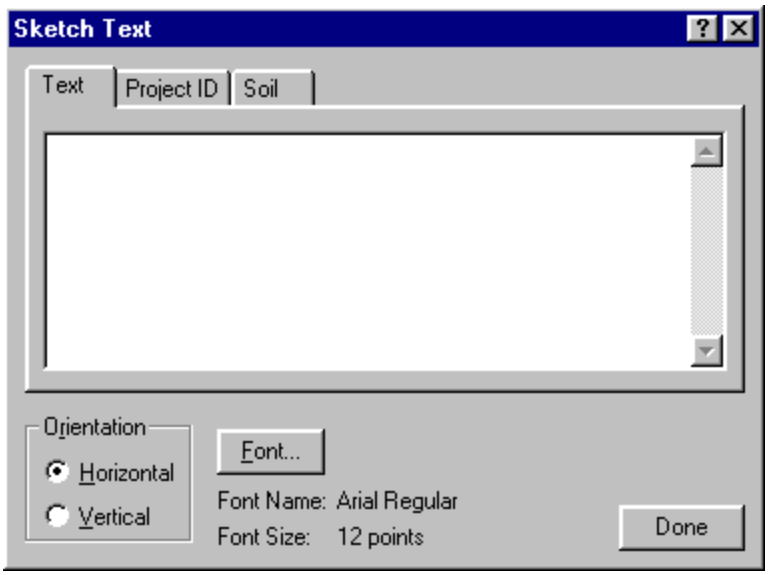
2. In the Title edit box, enter a title for this example problem, such as **SLOPE/W Example Problem**.

3. In the Comments edit box, enter a problem description, such as **Learn Example in Chapter 3**.

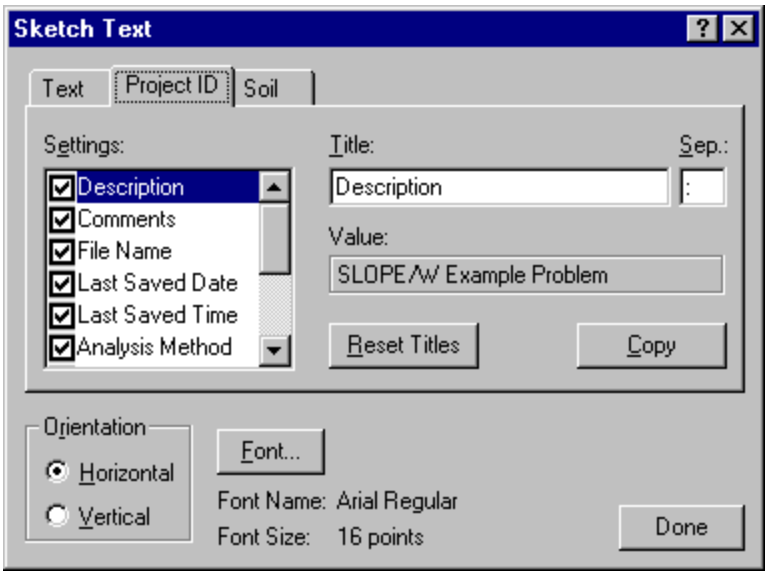
4. Press OK.

➢ **To place a Project ID text label on the drawing:**

1. Choose Text from the Sketch menu. The following dialog box appears:



2. Select the Project ID tab at the top of the dialog box. A Project ID property sheet is displayed in the dialog box:



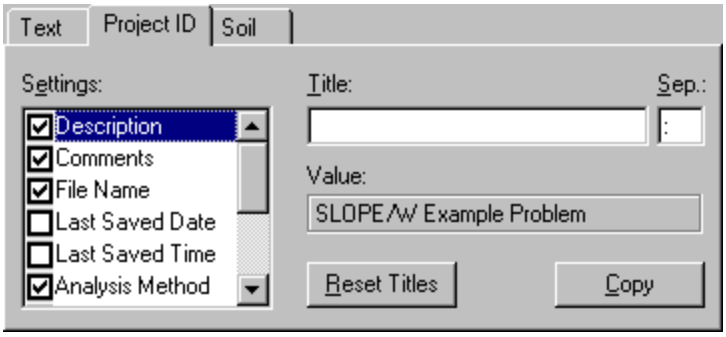
By default, all parameters are checked in the Settings list box.

3. In the Settings list box, check the parameters that you wish to include in the Project ID label. For example, uncheck all parameters except the Description, Comments, File Name and Analysis Method check boxes. (Be sure to use the scroll bar to view all of the parameters in the Settings list box.)

4. To remove the Description Title text, select Description in the list box, double-click the left mouse button inside the Title edit box and press the Delete key.

   Repeat this step for the Comments parameter to remove the Comments Title text.

   When you have completed the previous two steps, the Project ID property sheet should appear as follows (note that only the Description parameter is checked and it has no Title):



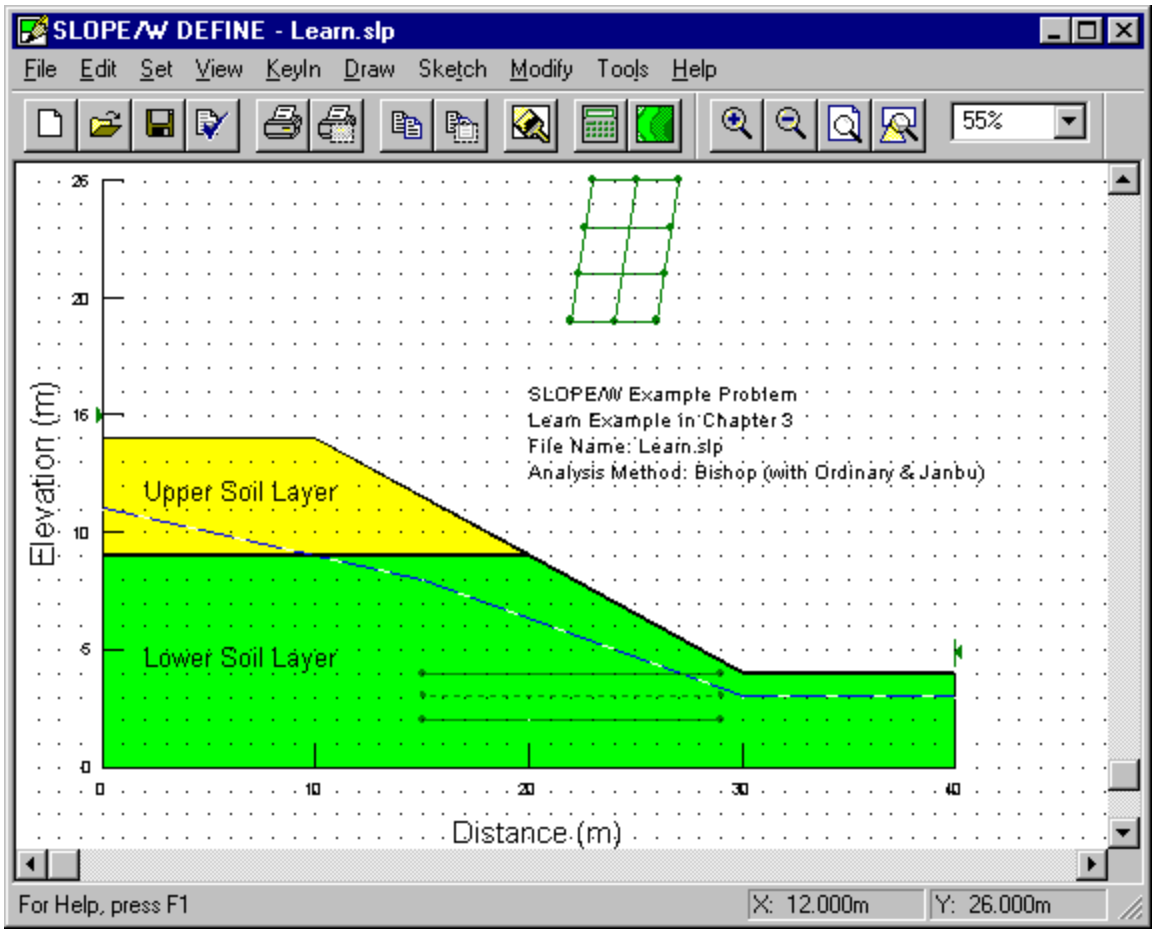
5. To place the Project ID label on the drawing, click the left mouse button near the (20,12) position in the DEFINE window.

   The label appears on the drawing above and to the right of the selected position.

6. Select Done to finish identifying the problem.

NOTE: If you change the project ID, file name, or analysis method, the Project ID label will be automatically updated to show the new settings. If you wish to display more of the project settings in the project label, choose the Modify Text command and click on the Project ID label.

After you have completed the above steps, your screen should look like the following:



Verify the Problem

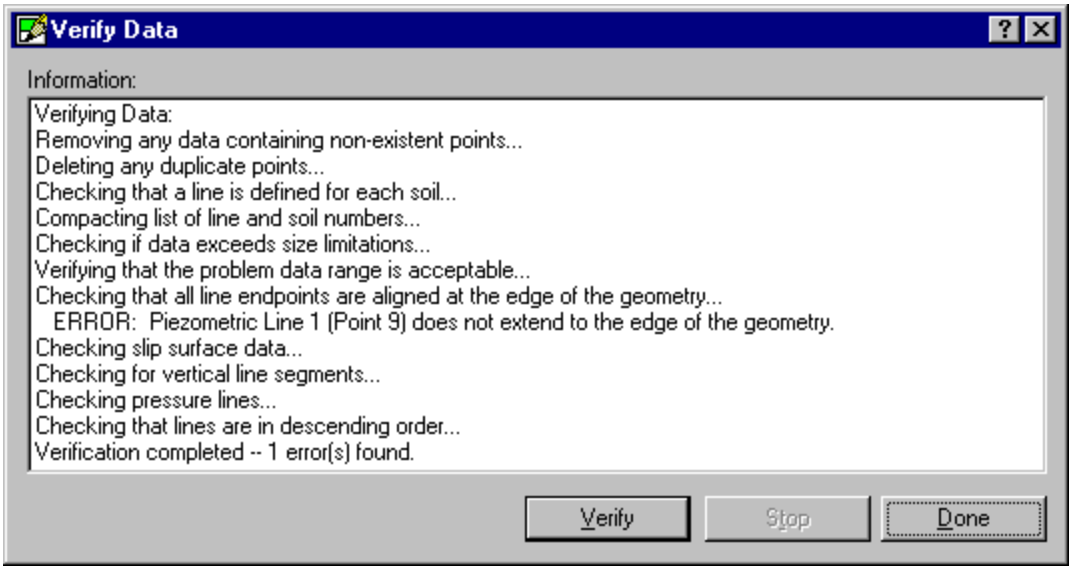
The problem definition should now be verified by SLOPE/W to ensure that the data has been defined correctly. The Tools Verify command performs a number of checks to help you find errors in the problem definition.

➢ **To verify the problem:**

1. Choose Verify from the Tools menu. The following dialog box appears:

2. Select the Verify button.

   SLOPE/W verifies the problem data. If any errors are found in the data, error messages are displayed in the dialog box. The total number of errors found is displayed as the last line in the dialog box. For example, if one of the endpoints in Piezometric Line 1 does not extend to the edge of the geometry, the following is displayed in the Verify Data dialog box:


Verify Data
Information:
Verifying Data:
Removing any data containing non-existent points...
Deleting any duplicate points...
Checking that a line is defined for each soil...
Compacting list of line and soil numbers...
Checking if data exceeds size limitations...
Verifying that the problem data range is acceptable...
Checking that all line endpoints are aligned at the edge of the geometry...
ERROR: Piezometric Line 1 (Point 9) does not extend to the edge of the geometry.
Checking slip surface data...
Checking for vertical line segments...
Checking pressure lines...
Checking that lines are in descending order...
Verification completed -- 1 error(s) found.
Verify
Stop
Done

3. To see all the verification messages in the list box, re-size the dialog box by dragging the bottom edge of the window down until all information is displayed.

4. When you are finished viewing the messages in the Verify Data dialog box, select Done.

Save the Problem

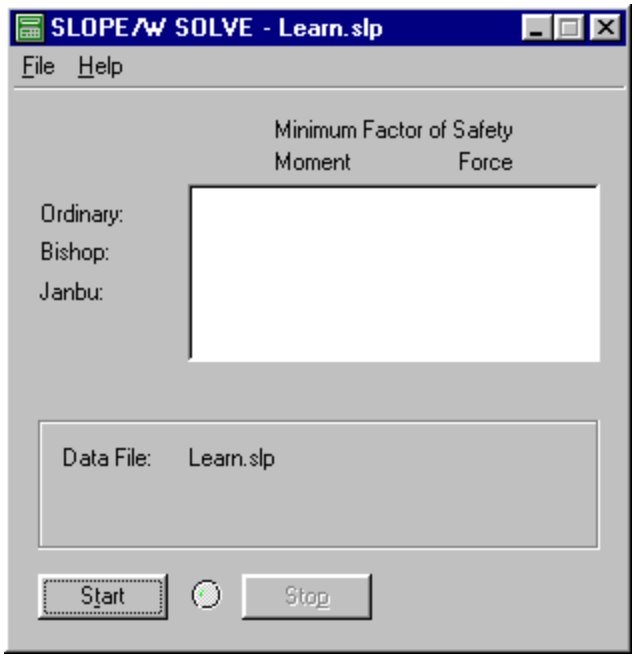
The problem definition is now complete. Choose File Save to save the problem definition to the same file name it was previously saved to, such as LEARN.SLP. SOLVE reads the problem data from this file to calculate the factors of safety.

Solving the Problem

The second part of an analysis is to use the SLOPE/W SOLVE function to compute the factors of safety.

To start SOLVE and automatically load the LEARN.SLP data file, click on the SOLVE button in the DEFINE Standard toolbar:

The SOLVE window appears. SOLVE automatically opens the LEARN.SLP data file and displays the data file name in the SOLVE window:



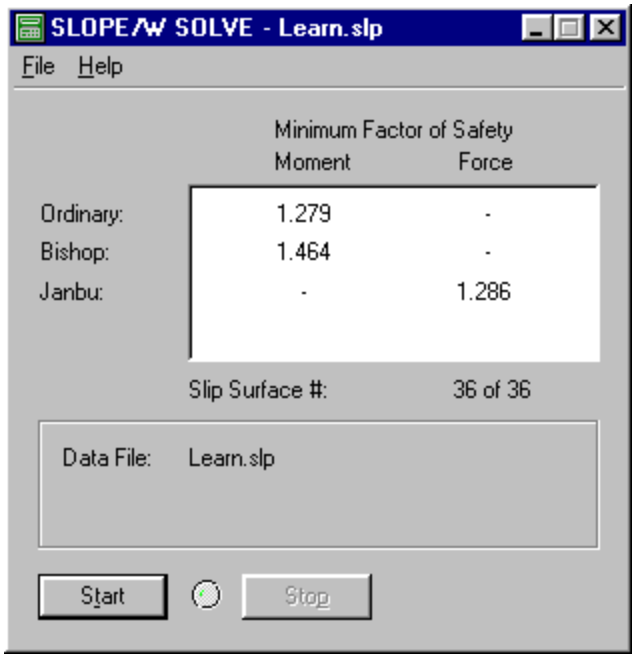
Alternatively, you can start SOLVE by clicking the SOLVE icon in the SLOPE/W Group folder and opening LEARN.SLP with the File Open Data File command. It is simpler, however, to start SOLVE from the DEFINE Standard toolbar when you wish to analyze a problem you have just defined. For more information about opening data files, see File Open Data File in Chapter 5.

Start Solving

To start solving for the factors of safety, click the Start button in the SOLVE window.

A green dot appears between the Start and Stop buttons; the dot flashes while the computations are in progress.

During the computations, SOLVE displays the minimum factors of safety and the number of the current slip surface being analyzed. For the example problem, a total of 36 slip surfaces are analyzed.



SOLVE writes the analysis results to a series of files, as described in the Limit Equilibrium Method section. CONTOUR reads these files in order to display the results.

Quit SOLVE

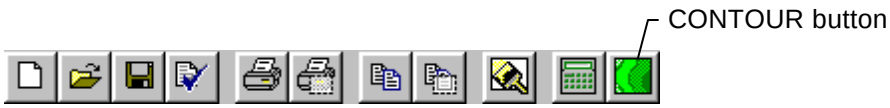
You have now computed the factors of safety. Choose File Exit to quit SLOPE/W SOLVE, or click the Minimize button in the top-right corner of the SOLVE window to reduce the window to an icon.

Viewing the Results

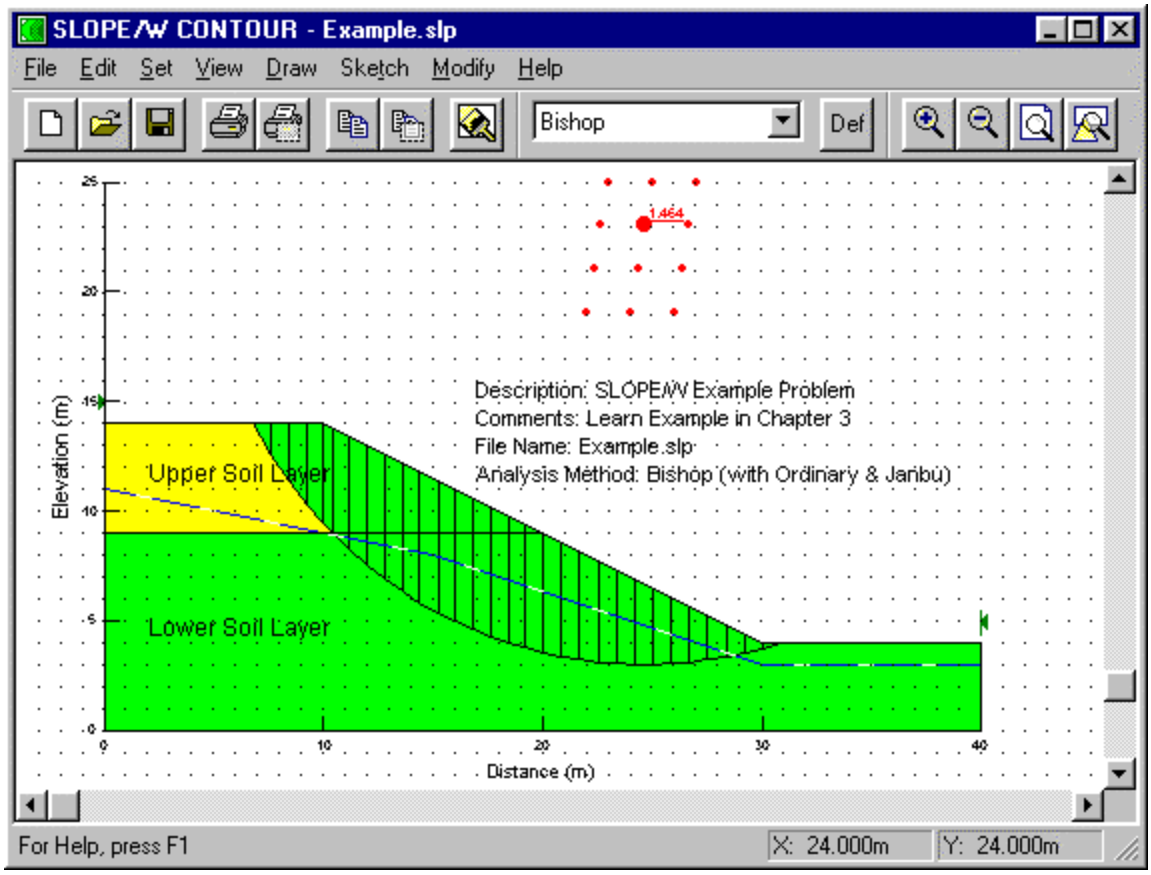
The SLOPE/W CONTOUR function allows you to view the results of the problem analysis graphically by:

- Displaying any of the analyzed slip surfaces, along with the associated factors of safety.
- Generating contour plots of the factors of safety.
- Displaying a free body diagram and force polygon for any slice in the minimum slip surface.
- Plotting graphs of the computed results.

To start CONTOUR and automatically load the LEARN.SLP data file, click on the CONTOUR button in the DEFINE Standard toolbar (if DEFINE still has the LEARN problem open). This is the same way in which SOLVE was launched previously.



The CONTOUR window appears. CONTOUR automatically opens the LEARN.SLP data file:



Alternatively, you can start CONTOUR by clicking the CONTOUR icon in the SLOPE/W Group folder and opening LEARN.SLP with the File Open command. It is simpler, however, to start CONTOUR from the DEFINE Standard toolbar when you wish to view the results of a problem that has already been analyzed. For more information about opening files in CONTOUR, see File Open in Chapter 6.

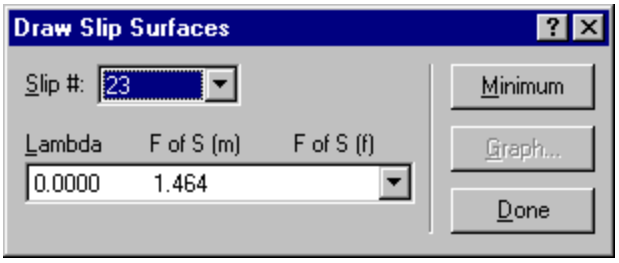
The drawing displayed in the CONTOUR window will be drawn according to the View Preferences selected at the time you saved the problem in DEFINE. You can view different parts of the drawing by choosing Preferences from the CONTOUR View menu or choosing items on the View Preference toolbar.

NOTE: You can select and unselect the View Preferences by clicking on the icons in the CONTOUR View Preferences toolbar. You can learn about each of the icons by placing the cursor over the icon. A tool tip will appear for a few seconds and a description is displayed on the status bar at the bottom of the window.

Draw Selected Slip Surfaces

➢ **To draw slip surfaces other than the minimum slip surface:**

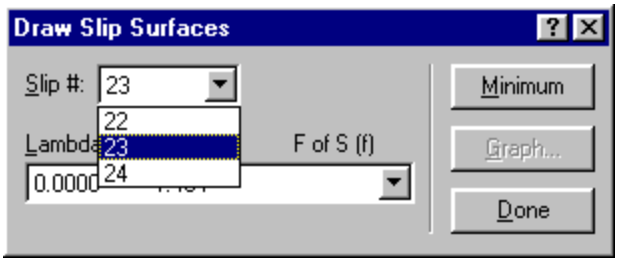
1. Choose Slip Surfaces from the Draw menu in CONTOUR. The following dialog box appears:



The cursor will change from an arrow to a cross-hair, and the status bar will indicate that "Draw Slip Surfaces" is the current operating mode.

The dialog box gives information about the slip surface currently displayed: the slip surface number, the Lambda value, and the moment and/or force factors of safety.

3. In the dialog box, click the down arrow to the right of the Slip # edit box. A drop-down menu of other slip surfaces with the same grid center is displayed.



4. Select any of the other slip surface numbers.

The selected slip surface and its factor of safety are displayed in the CONTOUR window. The dialog box displays the new slip surface number, its Lambda value, and its moment and/or force factors of safety.

5. To display another slip surface, move the cursor into the CONTOUR window near the grid rotation center of the desired slip surface and click the left mouse button.

CONTOUR draws the minimum slip surface for this center and displays the factor of safety beside the grid center point.

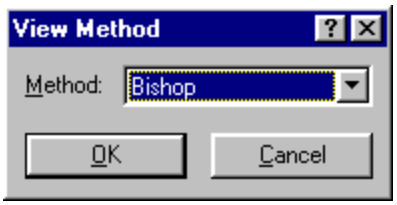
6. To finish viewing slip surfaces, select Done in the dialog box or click the right mouse button.

View Method

In DEFINE, you selected the Bishop (with Ordinary & Janbu) method to use when calculating the factors of safety. While CONTOUR displays the Bishop factors of safety by default, the Ordinary and Janbu factors of safety can also be viewed.

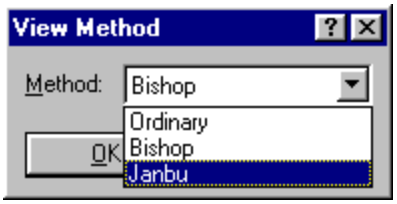
➢ **To view the factors of safety for another method:**

1. Choose Method from the View menu. The following dialog box appears:



The current method is displayed in the dialog box.

2. Click the down arrow to the right of the Method edit box. A drop-down menu of the other available methods to view is displayed.



3. Click on one of the other methods (e.g., Janbu).

4. Select OK.

Janbu is displayed in the Method Toolbar to indicate the currently viewed method. If the Method toolbar is not displayed, choose View Toolbars and select the Method checkbox.

The minimum slip surface computed for the Janbu method is displayed in the CONTOUR window; the Janbu factor of safety is displayed beside the grid center point. If you wish to view other slip surfaces for the Janbu method, choose Slip Surfaces from the Draw menu and select the slip surface to view.

NOTE: Instead of using the View Method command, you can select the method to view from the Method toolbar.

5. Choose View Method again and select Bishop to view the default method.

-- or –

Select the Default button from the Method toolbar.

The minimum slip surface for the default method (i.e., Bishop) is displayed in the CONTOUR window.

View the Slice Forces

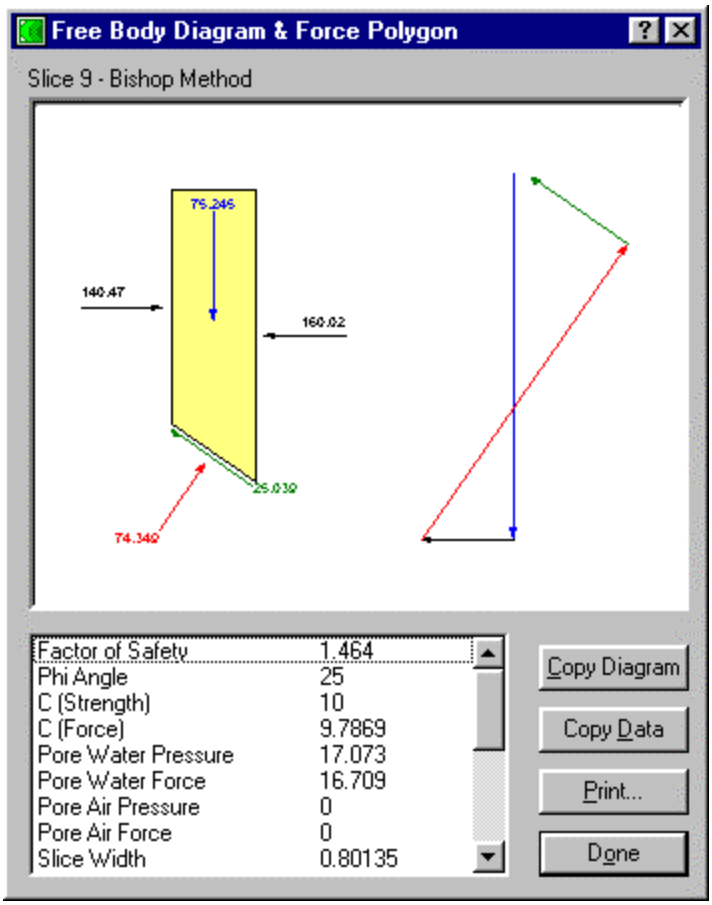
The forces calculated for the minimum slip surface can be displayed as a free body diagram and force polygon of any slice.

➢ **To view the slice forces:**

1. Choose Slice Forces from the View menu.

   An empty dialog box will appear. The cursor will change from an arrow to a cross-hair, and the status bar will indicate that "View Slice Forces" is the current operating mode.

2. Draw the forces on any slice by moving the cursor inside the slice and clicking the left mouse button. The following diagram appears in the window:



   The free body diagram shows the forces for the selected method on the minimum slip surface. The magnitude of each force vector is displayed beside the arrow (the length of the vectors is not drawn to scale), and the direction of the arrows represents the direction of the vectors. The force polygon shows the summation of all forces acting on the slice. Closure of the force polygon graphically represents the balance of the slice forces.

3. To enlarge the free-body diagram and force polygon, drag one of the window corners until the Slice Force Information window is the desired size.

4. Select Copy Diagram to copy the diagram to the Windows Clipboard for use in other Windows applications to create reports, slide presentations, or enhance the diagram.

5. Select Copy Data to copy the slice force information in the list box to the Windows clipboard in text format.

6. Select Print to print the diagram (at the size it is displayed on screen) and/or to print the slice force data.

7. Repeat Steps 2 to 6 until you have finished viewing slice force information. You can move the Slice Force Information window if you need to click on a slice that lies beneath the window.

8. Select Done or click the right mouse button to finish viewing slice forces.

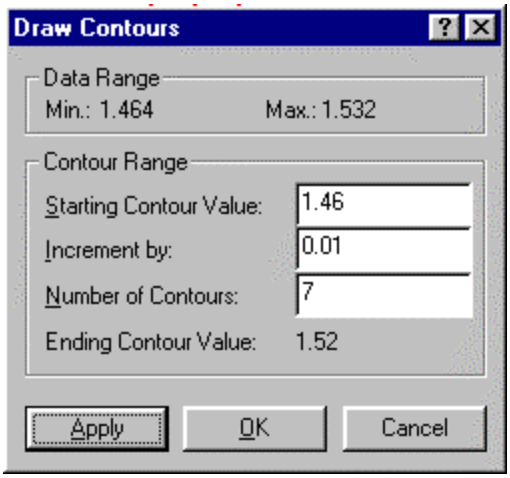
See View Slice Forces in Chapter 6 for further information on this command.

Draw the Contours

The minimum factors of safety at each of the grid centers can be contoured.

➢ **To contour the factors of safety:**

1. Choose Contours from the Draw menu. The following dialog box appears:



The Data group box displays the minimum and maximum factors of safety for the selected method. Default contour generation values are displayed in the edit boxes and can be used to contour the full range of factors of safety.

2. Type **0.01** in the Increment By edit box.

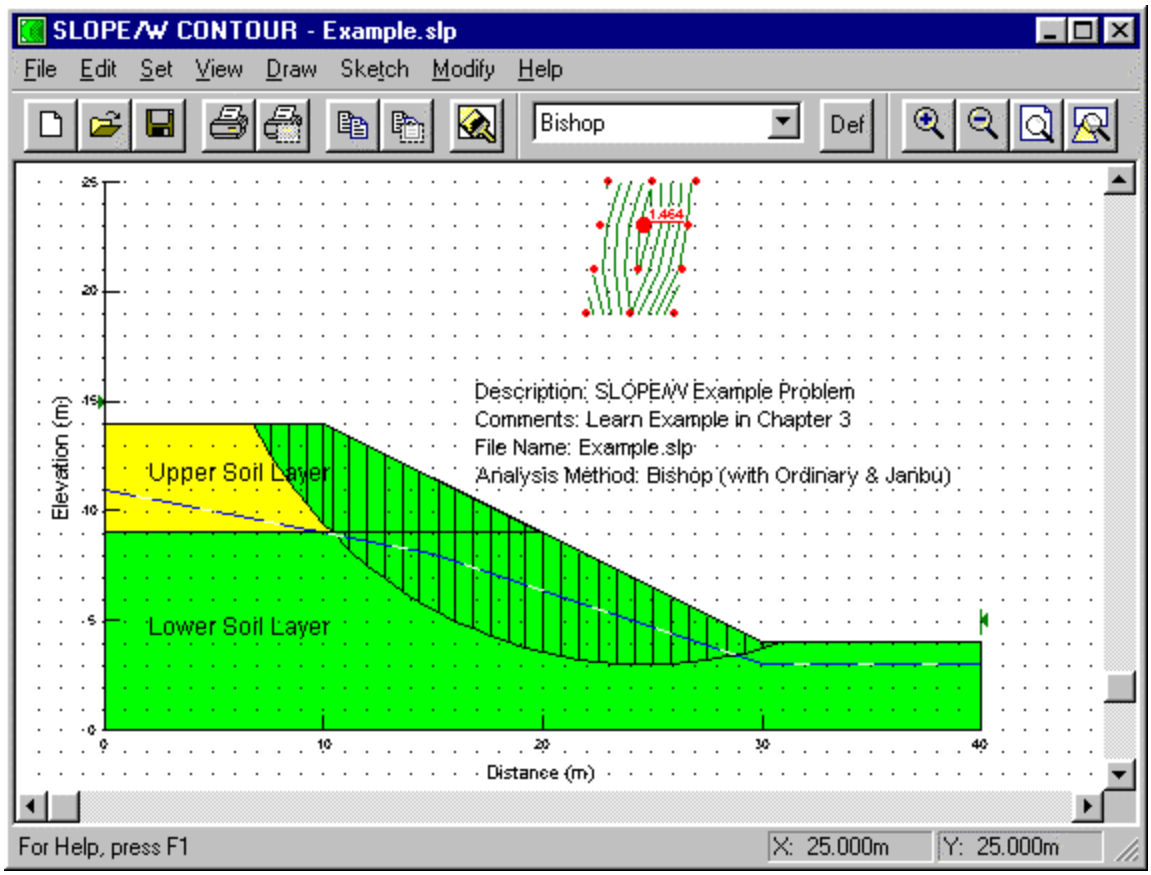
3. Type **7** in the Number Of Contours edit box.

4. Select Apply.

CONTOUR generates sequentially the specified number of contours in the list box. Repeat Step 2 if you wish to modify these contour values.

5. Select OK.

The factors of safety are contoured as follows:



Draw the Contour Labels

➢ **To label the contours on the drawing:**

1. Choose Contour Labels from the Draw menu.

   The cursor will change from an arrow to a cross-hair, and the status bar will indicate that “Draw Contour Labels” is the current operating mode.

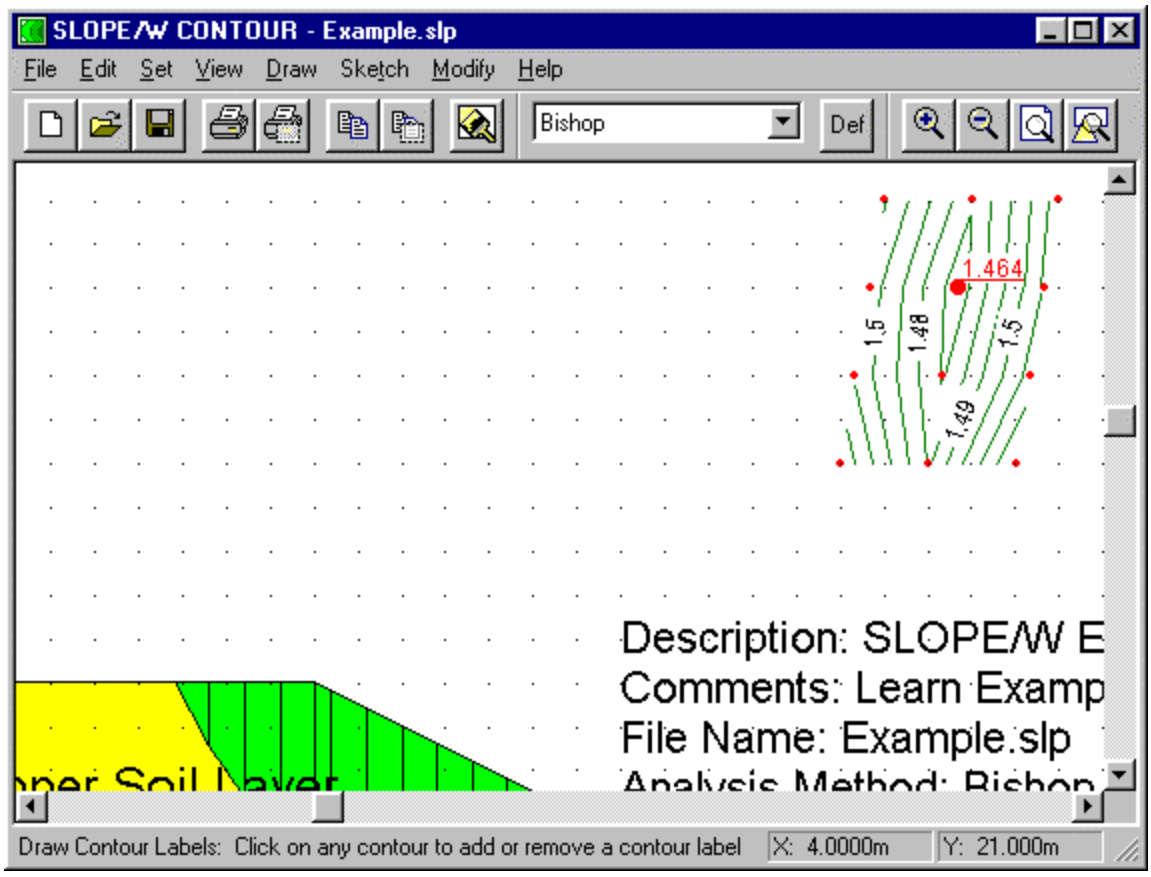
2. Move the cursor to a convenient point on a contour, and click the left mouse button.

   The contour value appears on the contour. If you wish to remove the contour label, simply re-click on the label, and the label disappears. Click again, and the label will re-appear.

3. Repeat Step 2 for as many contours as you wish.

4. Press ESC or click the right mouse button to finish labeling the contours.

After the contours are labelled, the factors of safety grid should look similar to the following:

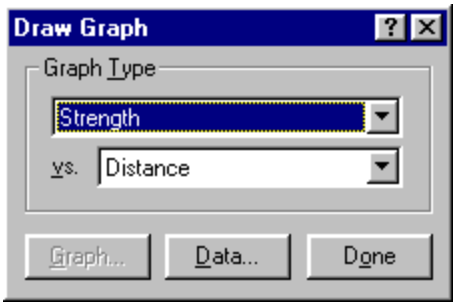


Plot a Graph of the Results

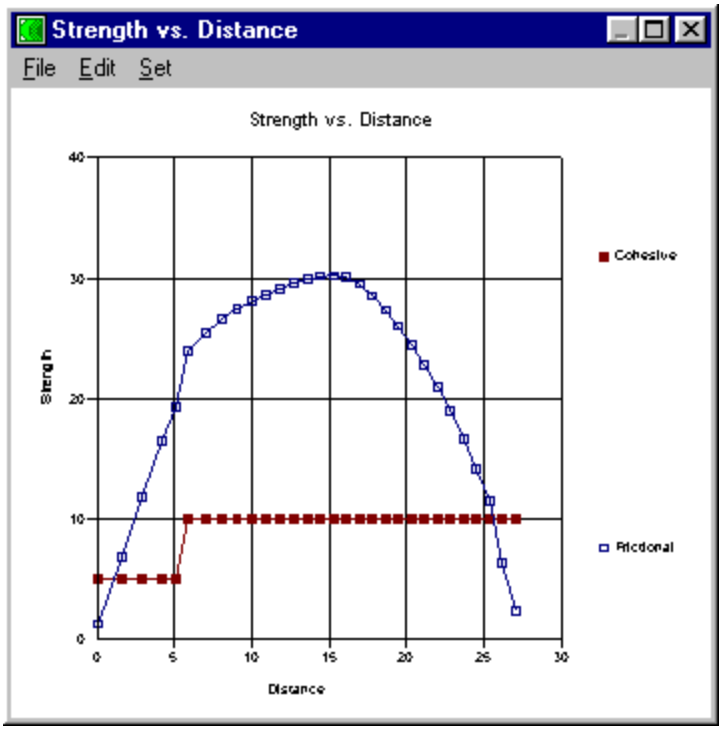
The forces acting on each slice for the critical slip surface are computed and saved in a file with a file name extension of FRC. While CONTOUR allows you to display a free body diagram of these forces, you can also view graphs of these forces. For this example problem, the procedures will be presented for plotting the pore-water pressure distribution from crest to toe along the critical slip surface.

➢ **To plot the graph:**

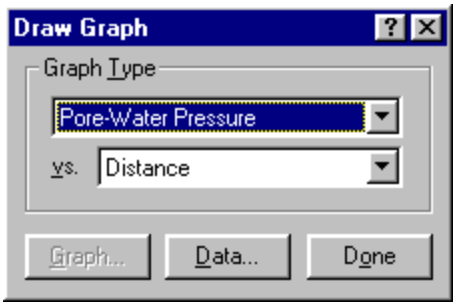
1. Choose Graph from the Draw menu. The following dialog box appears:



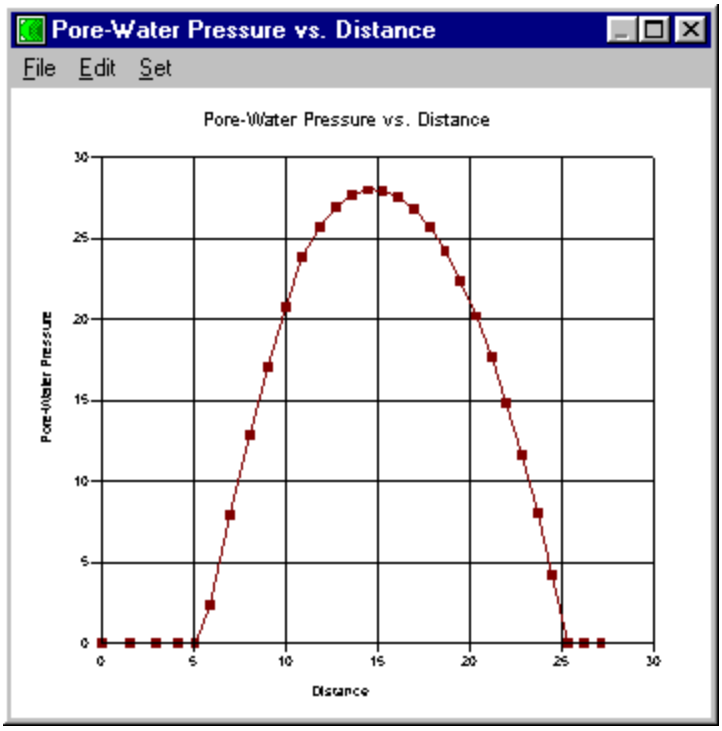
The following Graph window also appears, containing a graph of the selected conditions:



2. Click on the down arrow to the right of the first drop-down list box. A drop-down list of the other available conditions to plot is displayed.



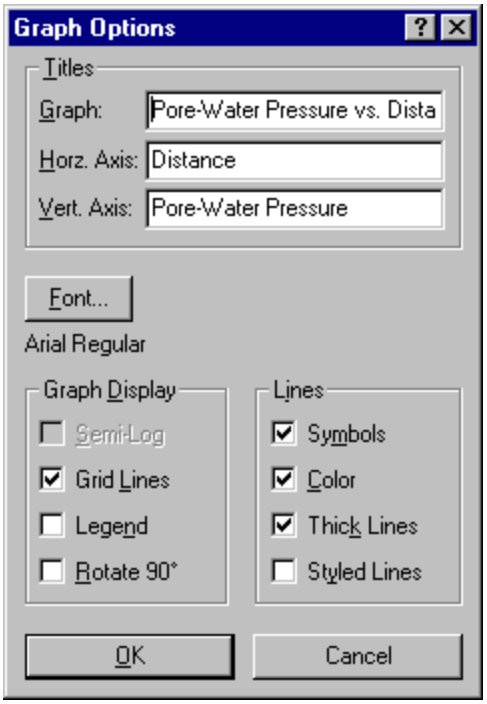
3. Select Pore-Water Pressure from the drop-down list. The following graph is displayed:



4. Repeat Steps 2 to 3 for any other graphs that you wish to display.

5. Select File Print from the Graph window menu if you wish to print the graph on the default printer. Select Edit Copy from the Graph window menu if you wish to copy the graph to the Windows Clipboard for importing into other applications.

6. Select Set Options to specify the titles and display options of the graph. The following dialog box is displayed:



7. Click on the Font button to specify the font style of the graph.

8. To close the Graph window, double-click on the control-menu box in the upper-left corner of the Graph window.

9. Select Done from the Draw Graph dialog box.

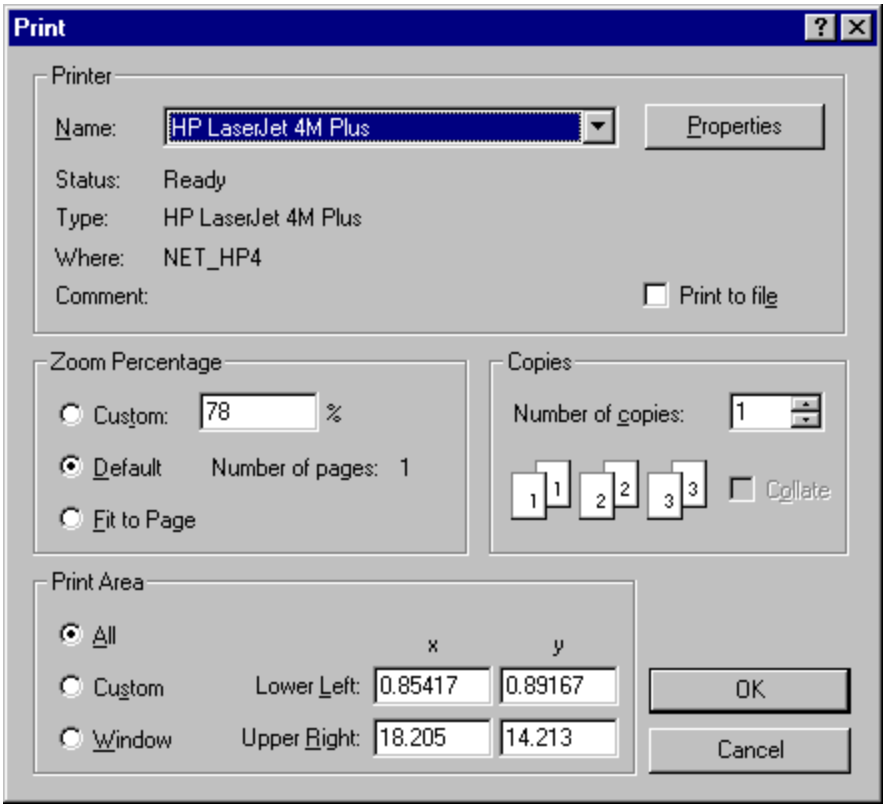
See the Draw Graph command reference in Chapter 6 for a complete discussion of the CONTOUR graphing capabilities, since there are other features of the command that have not been discussed in this section.

Print the Drawing

➢ **To print the CONTOUR drawing:**

1. Ensure that the entire drawing is displayed in the window before printing. To display the entire drawing in the window click on the Zoom Objects button in the Zoom toolbar. (If the Zoom toolbar is not displayed, choose View Toolbars and click on the Zoom check box).

2. Click on the Print button in the Standard Toolbar. The following dialog box appears:



3. Select OK to print the drawing on the default printer at the currently displayed size. For more information on printing, see the File Print command in Chapter 4.

You have now finished viewing the results. Choose File Exit to quit SLOPE/W CONTOUR, or click the Minimize button in the top-right corner of the CONTOUR window to reduce the window to an icon.

You have reached the end of this introductory learning session. You have learned sufficient concepts to give you a general understanding of the operation and capability of SLOPE/W. Not all of the powerful features of SLOPE/W have been used in this introductory learning session, nor have all of the technical details been discussed about the features that have been used. Specific details about each command are given in the chapters that follow.

The next section of this chapter will introduce some of the more advanced features available in SLOPE/W Version 4.

Using Advanced Features of SLOPE/W

This section illustrates how to use several advanced features that are available in SLOPE/W, including importing pictures, specifying a rigorous method of analysis and performing a probabilistic analysis.

To demonstrate these features we will make use of the LEARN.SLP example problem that was created in the introductory section of this chapter.

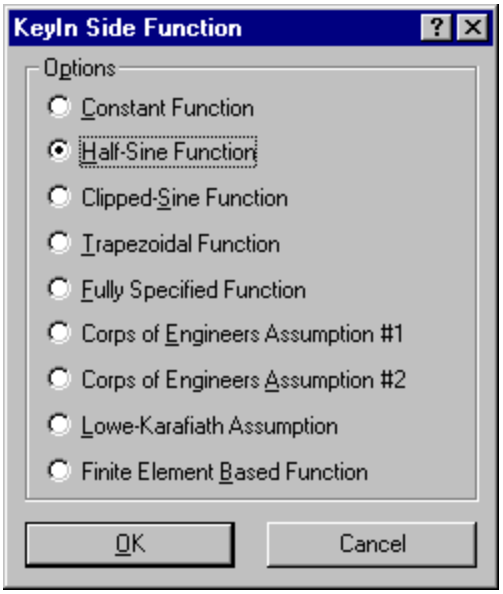
Specify a Rigorous Method of Analysis

SLOPE/W can compute the factor of safety for many methods. A question often asked is, "Which method gives the best value?" While there is no single answer to this question, the Adopting A Method section in Chapter 7 explains why specifying a rigorous method of analysis (e.g., Spencer, Morgenstern-Price or GLE) can result in a more accurate factor of safety. For this example problem, we will change the method of analysis from Bishop's Simplified to the rigorous Morgenstern-Price method.

➢ **To specify the use of a Rigorous Method of Analysis:**

1. Choose Analysis Method from the KeyIn menu and select the Morgenstern-Price method, as shown below:

2. Select OK. A Side Function dialog box is displayed, allowing you to select the function that will be used to compute the interslice shear forces:



3. Select the Half-Sine Function and then select the OK button.

See the Interslice Forces section in Chapter 8 for more information on selecting interslice shear force functions.

The method of analysis is now changed from Bishop's Simplified method to the rigorous Morgenstern-Price method.

Perform a Probabilistic Analysis

Deterministic slope stability analyses (such as the LEARN.SLP problem you have just analyzed) compute the factor of safety based on a fixed set of conditions and material parameters. In a deterministic analysis, there is no way of considering variability in the soil properties. A SLOPE/W probabilistic analysis allows you to consider the variability of input parameters (including soil properties).

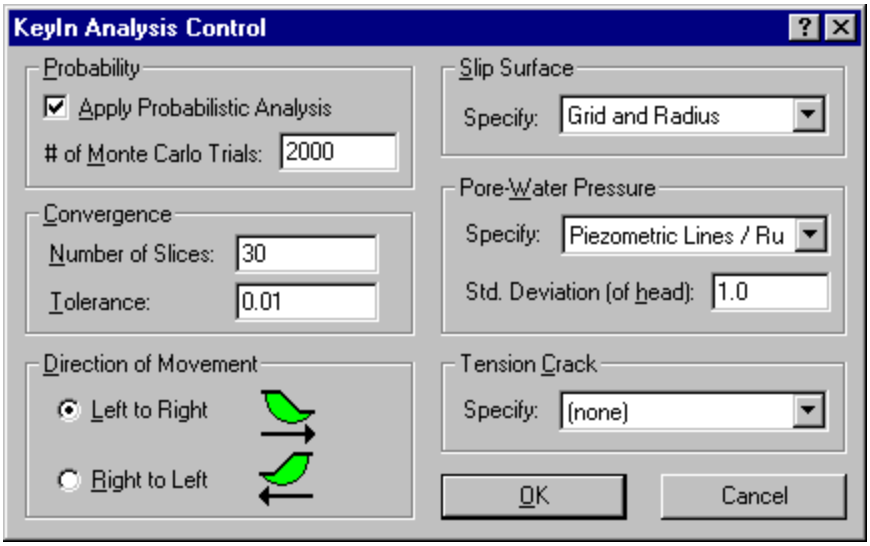
A probabilistic analysis also quantifies the probability of failure of a slope, making it possible for you to consider, "How stable is the slope?" A deterministic analysis cannot answer this question, since a slope is considered to be stable if the factor of safety is greater than unity or unstable if the factor of safety is less than unity.

SLOPE/W Version 4 performs probabilistic slope stability analyses using the Monte Carlo method. See Probabilistic Analysis in Chapter 7 and Probabilistic Slope Stability Analysis in Chapter 8 for further discussion on how SLOPE/W performs probabilistic analyses.

For this example problem, we will add a standard deviation to the soil properties and the piezometric line that you entered for LEARN.SLP.

➢ **To specify a probabilistic analysis in DEFINE:**

1. Choose Analysis Control from the KeyIn menu. The following dialog box appears:

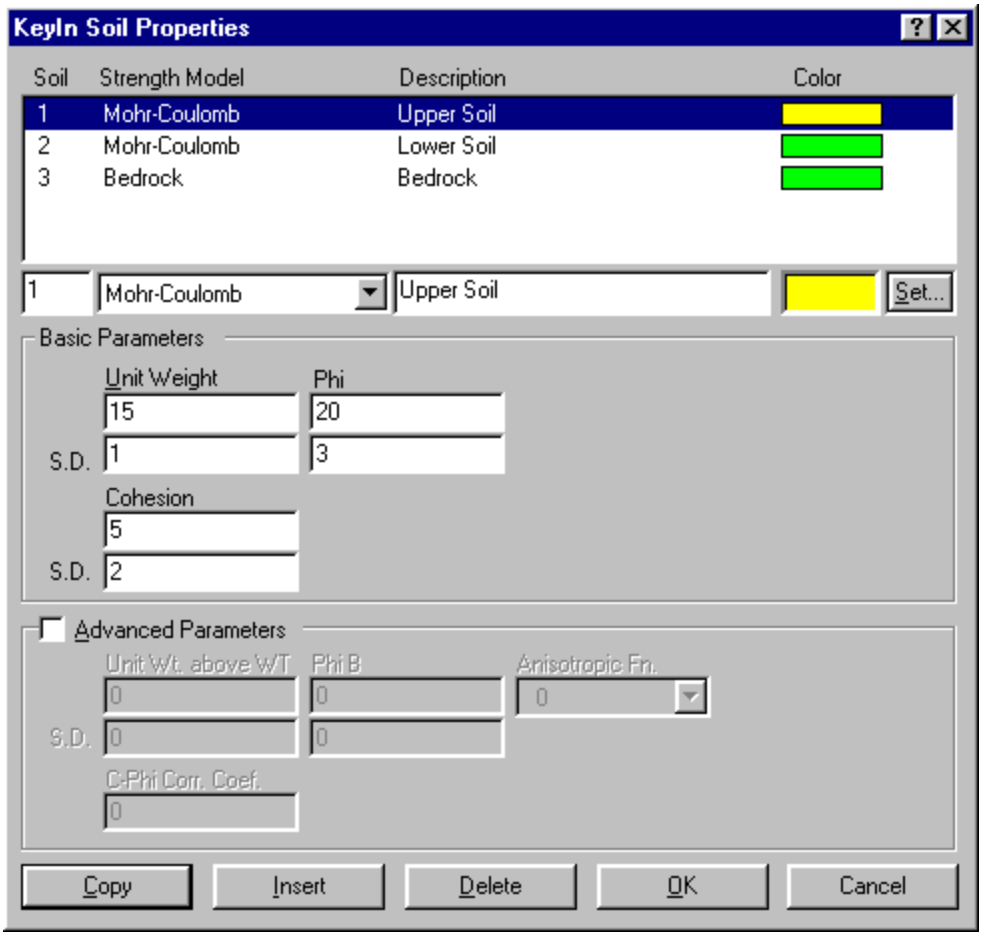


2. Check the Apply Probabilistic Analysis check box.
3. Type **2000** in the number of Monte Carlo Trials edit box.
4. Type **1** in the Std. Deviation (of head) edit box.

   The other values can remain the same from the original example problem
5. Click OK to apply the changes to the problem.

➢ **To add a standard deviation to the soil properties:**

1. Select Soil Properties from the KeyIn menu. The following dialog box appears:

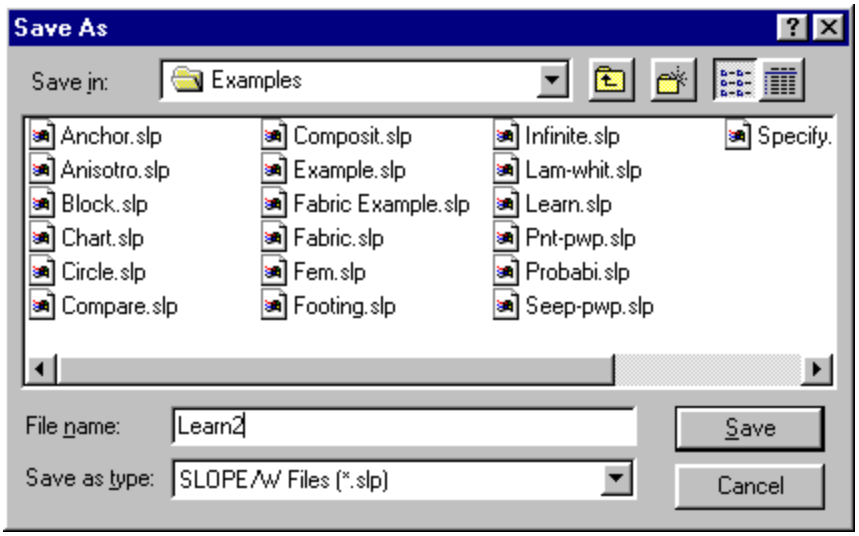


2. Select Soil 1 in the Soil Properties list box.
3. Type the following values for Soil 1 in the appropriate edit boxes:
   - Unit Weight: **15** Standard Deviation: **1**
   - Cohesion: **5** Standard Deviation: **2**
   - Phi: **20** Standard Deviation: **3**
4. Click Copy to apply the changes you have made to Soil 1.
5. Select Soil 2 and enter the following values in the appropriate edit boxes:
   - Unit Weight: **18** Standard Deviation: **2**
   - Cohesion: **10** Standard Deviation: **2**
   - Phi: **25** Standard Deviation: **5**
6. Click Copy to apply the changes you have made to Soil 2.

7. Select OK.

➢ **To save the probabilistic analysis:**

1. Choose Save As from the File menu to save the new data file under a different file name. The following dialog box appears:



2. Type **LEARN2** in the File name edit box.

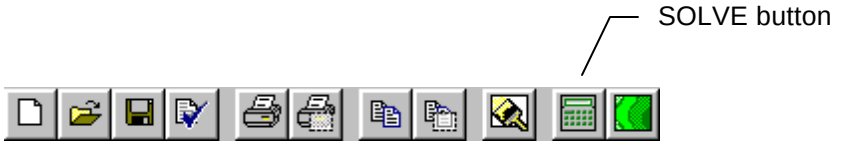
3. Click on the Save button.

   The file is saved as LEARN2.SLP.

NOTE: Although not considered in this example problem, variation in the line load magnitudes and seismic coefficients can also be considered in SLOPE/W probabilistic analysis. See Probabilistic Slope Stability Analysis in Chapter 8 for more information.

➢ **To solve the probabilistic analysis:**

Start SOLVE by clicking on the SOLVE button in the Standard toolbar (if DEFINE still has the LEARN2 problem open). This will automatically load the LEARN2.SLP data file:



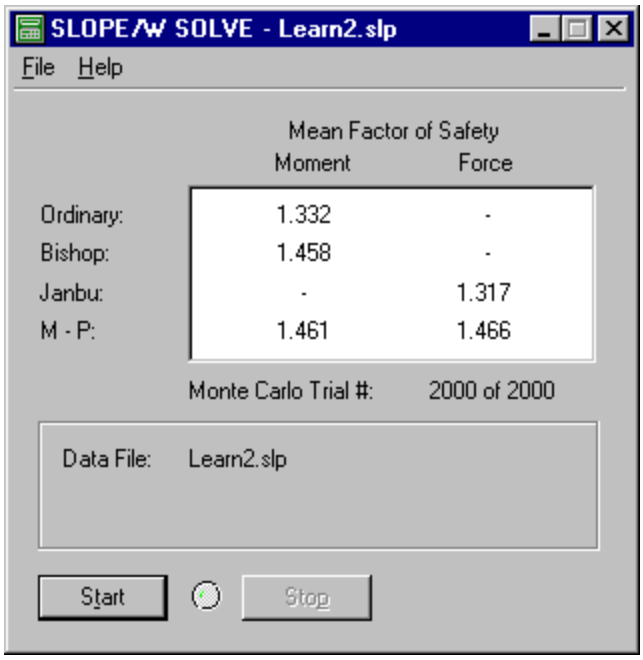
The SOLVE window appears. SOLVE automatically opens the LEARN2.SLP data file and displays the data file name in the SOLVE window.

To start solving for the factors of safety, click the Start button in the SOLVE window.

A green dot appears between the Start and Stop buttons; the dot flashes while the computations are in progress.

During the probabilistic analysis, the minimum factors of safety obtained using the mean input parameters (i.e., without variability) for the different methods are displayed.

When the probabilistic analysis is complete, the mean factors of safety at the critical slip surfaces are displayed for the different methods, including Morgenstern-Price (M-P):

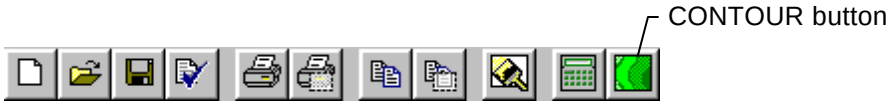


NOTE: The mean factor of safety will be different each time that you run SOLVE. The amount of difference depends on the degree of variability in the input parameters and the number of Monte Carlo trials used for the analysis. If the mean factor of safety varies considerably each time you run the analysis, you may want to increase the number of Monte Carlo trials. See the Monte Carlo Method section in Chapter 8 for more information.

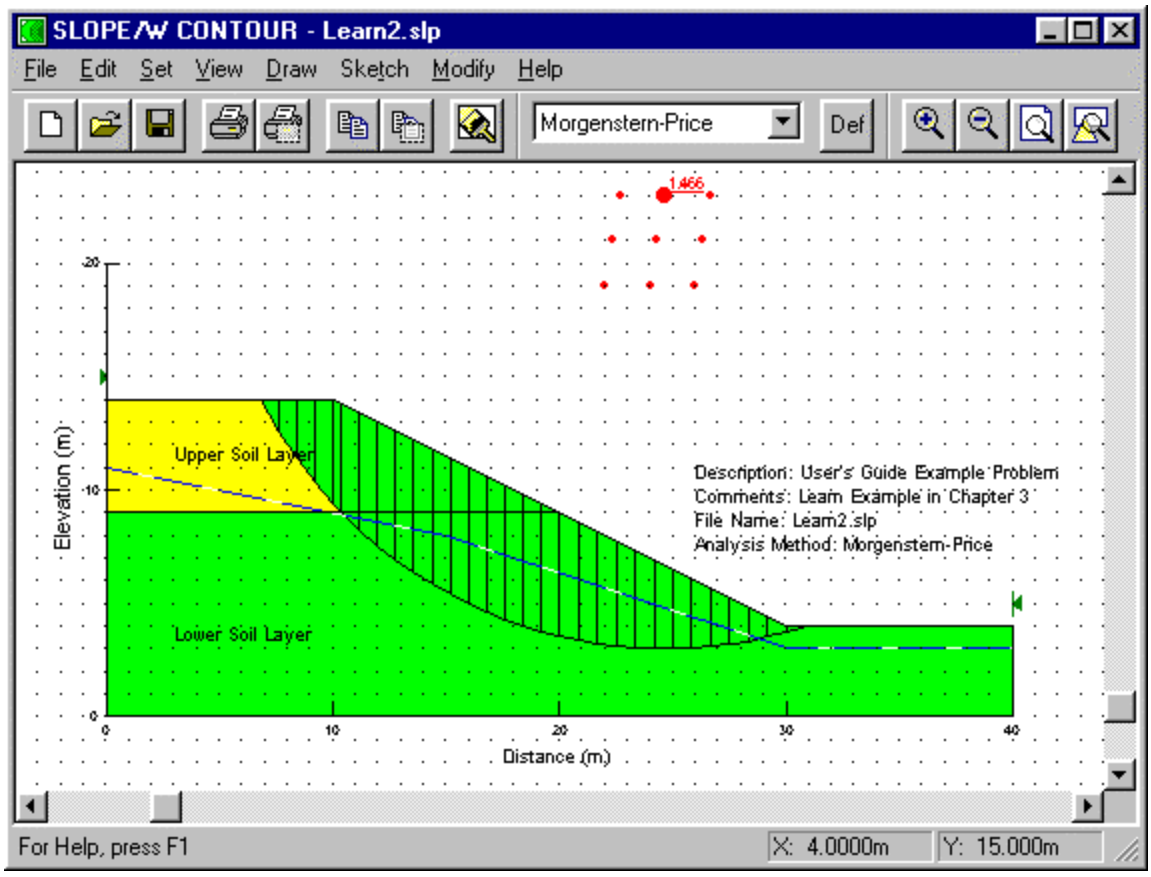
You have now computed the factors of safety. Choose Exit from the File menu to quit SLOPE/W SOLVE, or click the Minimize button in the top-right corner of the SOLVE window to reduce the window to an icon.

➢ **To view the probabilistic analysis results in CONTOUR:**

Start CONTOUR by clicking on the CONTOUR button in the Standard toolbar (if DEFINE still has the LEARN2 problem open). This will automatically load the LEARN2.SLP data file in the same way that SOLVE was launched previously:



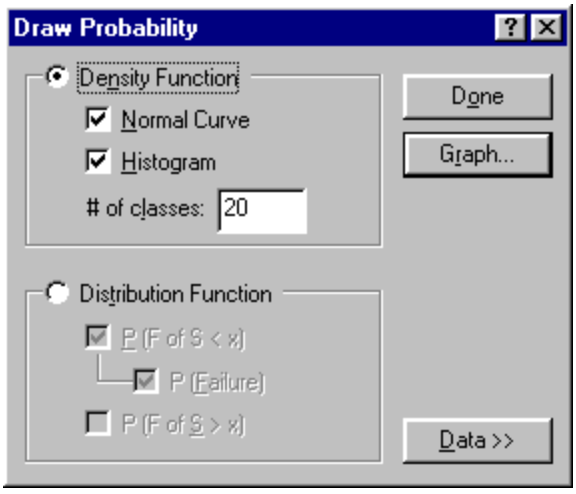
The CONTOUR window appears. CONTOUR automatically opens the LEARN2.SLP data file:



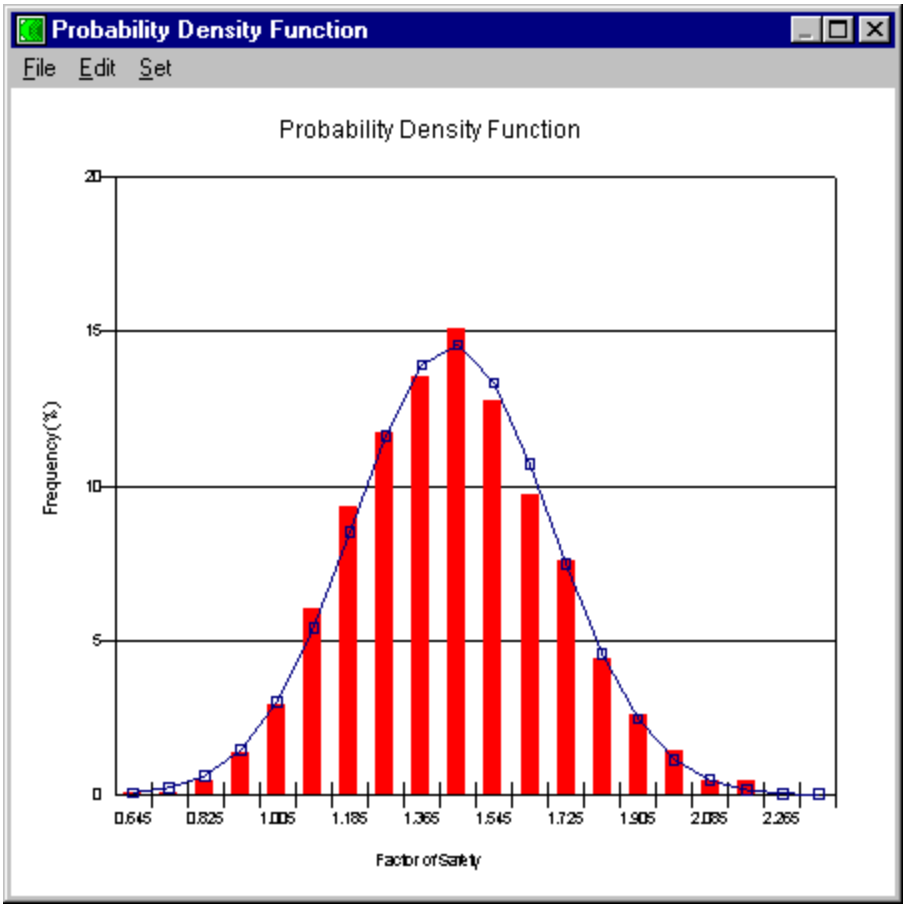
NOTE: The factor of safety displayed on the grid center is always the minimum factor of safety using the mean input parameters. It is not the mean factor of safety for all the Monte Carlo trials.

➢ **To graph the probabilistic analysis results:**

1. Choose Probability from the CONTOUR Draw menu. The following dialog box appears:

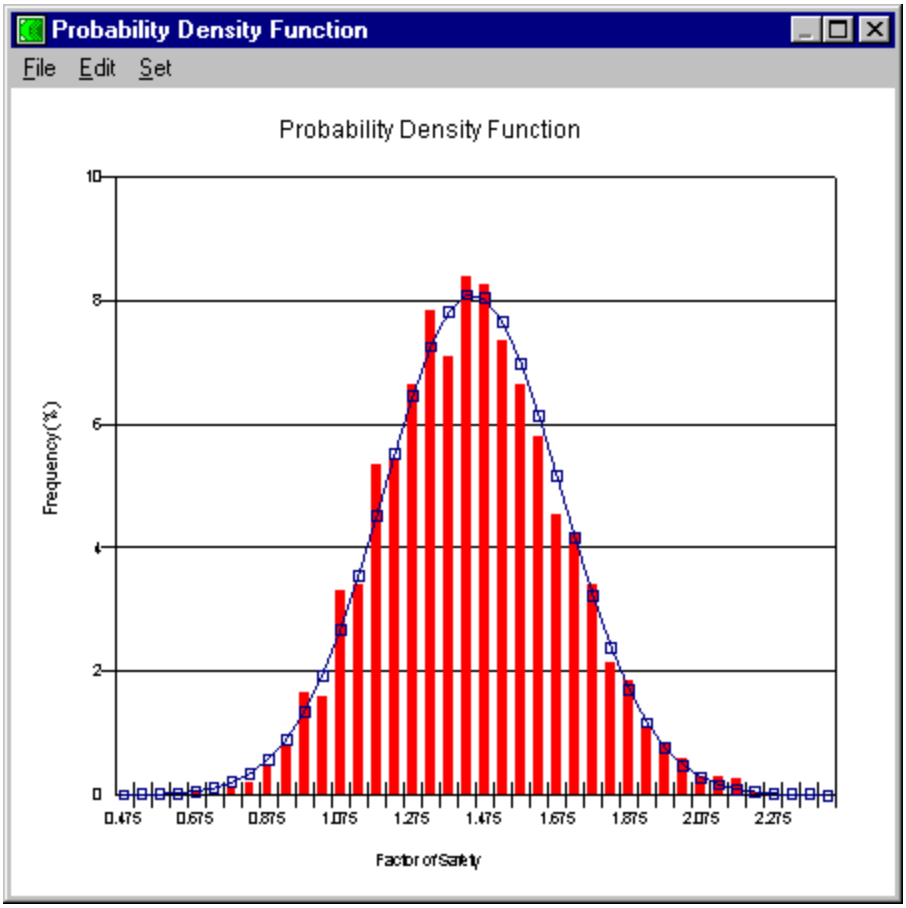


By default, a Probability Density Function graph is displayed showing the normal curve. The histogram of the factors of safety are also displayed:

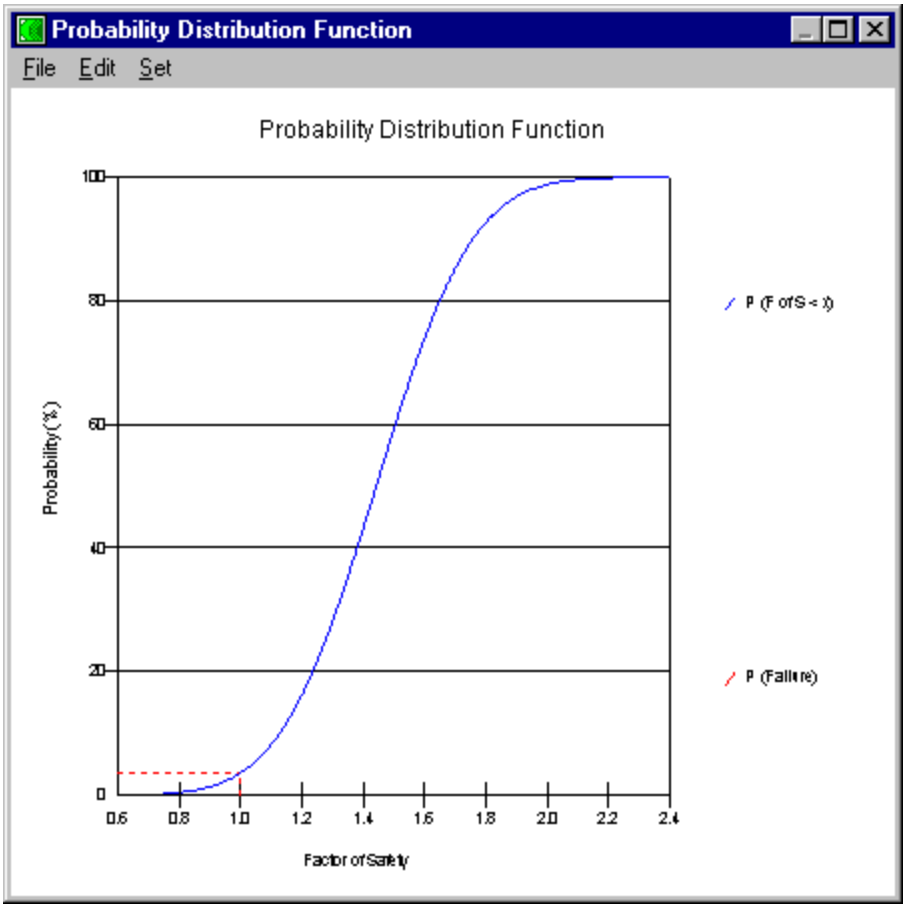


The Frequency (%) shows the distribution of the Monte Carlo trial factors of safety in terms of percentage. By default, the number of classes is assumed to be **20**.

2. Change the number of classes to **40** in the # of classes edit box and click on the Refresh button. The Graph window is updated and the following Probability Density Function is displayed:

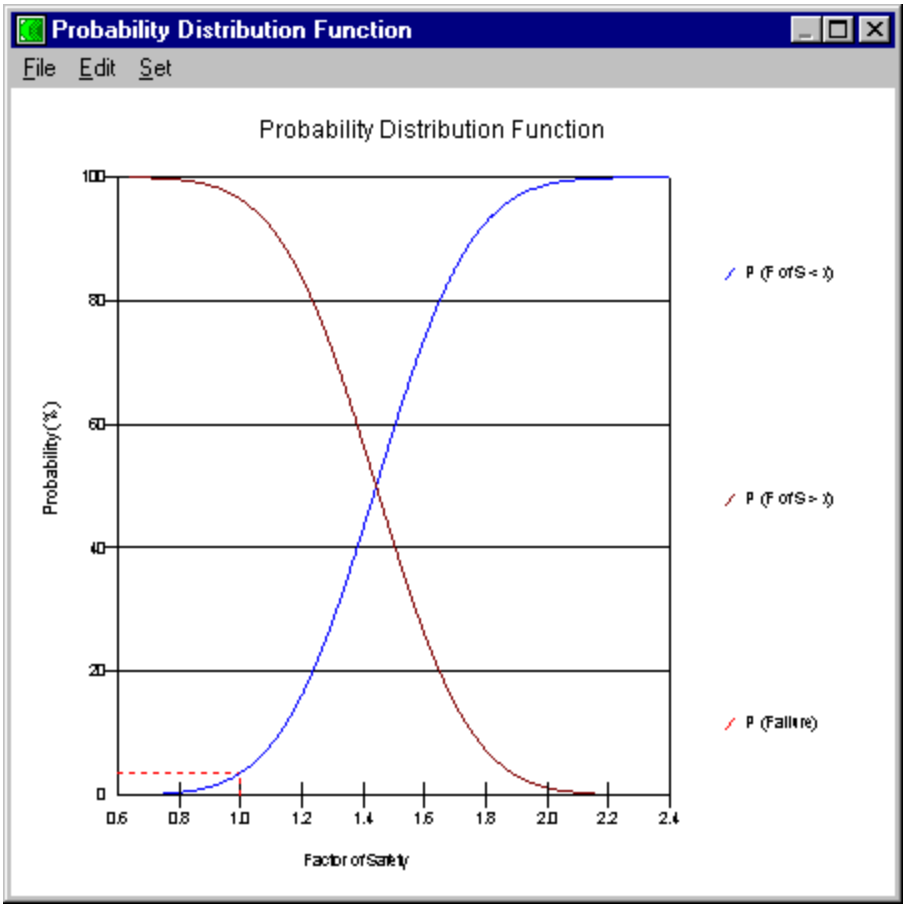


3. View the Probability Distribution Function by selecting the Distribution Function button in the Draw Probability dialog box. The following Probability Distribution Function is displayed:

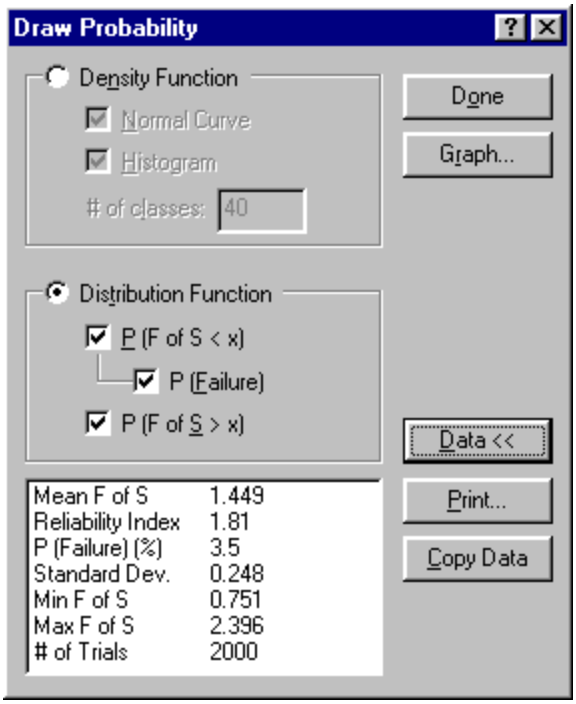


The above function is the Probability Distribution Function for factors of safety less than any given factor of safety. The dotted red line shows the probability that the factor of safety will be less than 1.0 (i.e., the probability of failure).

4. The Probability Distribution Function for factors of safety greater than any given factor of safety can a be viewed by selecting the corresponding button in the Draw Probability dialog box.



5. Select the Data<< button from the Draw Probability dialog box to show the various probability results such as the mean factor of safety, the reliability index and the probability of failure:



6. Select File Print from the Graph window menu if you wish to print the graph on the default printer. Select Edit Copy from the Graph window menu if you wish to copy the graph to the Windows Clipboard for importing into other applications.

7. Select Done to close the Probabilistic Data graph and the Draw Probability window.

See Draw Probability in Chapter 6 for more information on this command.

Import a Picture

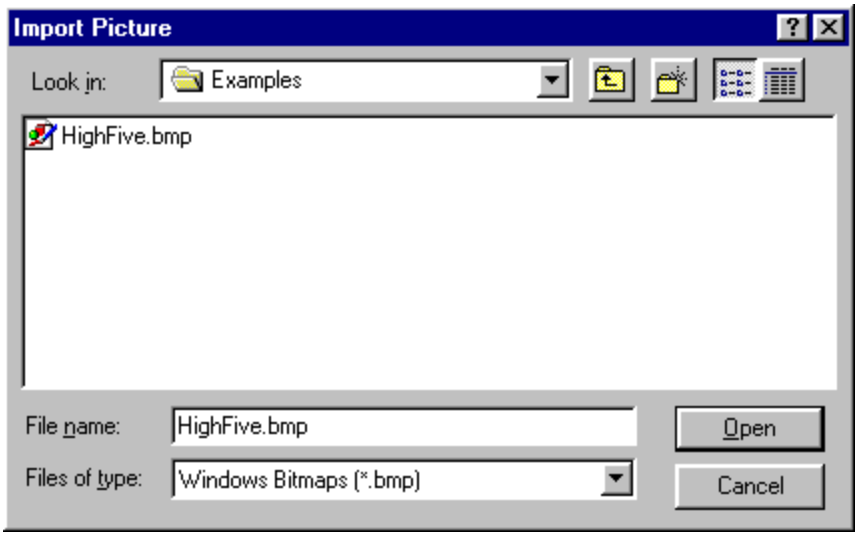
The SLOPE/W Import Picture command is useful if you wish to enhance your SLOPE/W drawing with a picture that you have created with another Windows program. For example, you may wish to insert a company logo, photograph, or other image into your SLOPE/W drawing. You can also use the Import Picture command to import a previously-defined cross-section into SLOPE/W and use it as a background for drawing your SLOPE/W geometry.

In this example, we will use the Import Picture command to import a corporate logo into the LEARN2.SLP problem.

➢ **To import a picture into the problem:**

1. Start DEFINE and open the LEARN2.SLP problem that you created earlier.

2. Choose Import Picture from the File menu. The following dialog box appears:



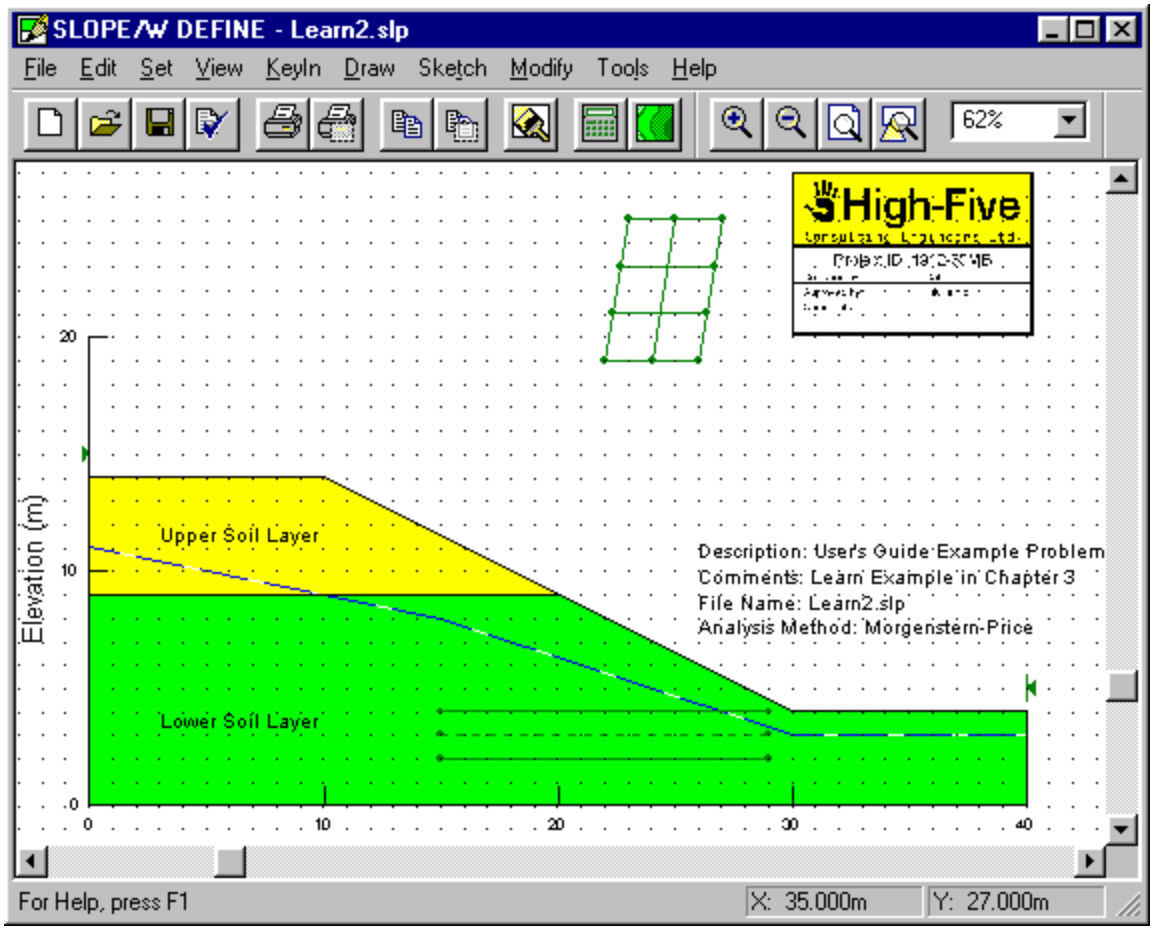
3. Select the bitmap file **HighFive.bmp** and click Open.

   The Import Picture dialog box disappears, the cursor changes from an arrow to a cross-hair, and the status bar indicates that "Import Picture" is the current operating mode.

4. Move the cursor to the position on the drawing where you wish to place the imported picture, such as (30,22), and click the left mouse button.

   The picture is placed on the drawing such that the bottom-left corner is aligned with the cursor position.

After you have placed the logo in the drawing, your screen should look like the following:



➢ **To change the size or position of the imported logo:**

1. Choose the Modify Objects command from either the Modify menu or from the Mode toolbar.

   The cursor changes from a white arrow to a black arrow, the status bar indicates that "Modify Objects" is the current operating mode, and the Modify Objects dialog box appears.

2. In the DEFINE window, click on the logo graphic using the left mouse button.

3. Move the graphic by dragging the object with the mouse to a suitable position on the drawing.

4. Select Done or press the ESC key to finish modifying objects.

NOTE: Multiple pictures may be imported into a single drawing. For example, you can use the Import Picture command to place a background picture of a slope, a company logo and a standard company template all on the same drawing. You can also use the Modify Pictures command to control the display order of multiple pictures and scale imported pictures to that of the slope in the drawing.

You have reached the end of this advanced learning session. The two example problems created in this chapter (LEARN.SLP and LEARN2.SLP) are included as EXAMPLE.SLP and EXAMPLE2.SLP in the SLOPE/W examples directory.

Additional illustrative examples can be found in Chapter 7 and Chapter 9; these examples further describe the various capabilities and features of SLOPE/W.

Chapter 4 DEFINE Reference

Introduction 3

Toolbars 4
Standard Toolbar 4
Print Selection Button 5
Copy Selection Button 6
Mode Toolbar 7
View Preferences Toolbar 9
Grid Toolbar 10
Zoom Toolbar 11

The File Menu 13
File New 13
File Open 15
File Import: Data File 16
File Import: Picture 17
File Export 19
File Save As 20
File Print 22
File Save Default Settings 23

The Edit Menu 25
Edit Copy All 25

The Set Menu 26
Set Page 26
Set Scale 28
Set Grid 30
Set Zoom 31
Set Axes 31

The View Menu 34
View Point Information 34
View Soil Properties 35
View Preferences 37
View Toolbars 40
View Redraw 41

The KeyIn Menu 42
KeyIn Project ID 43
KeyIn Analysis Method 45
KeyIn Analysis Control 54
KeyIn Soil Properties 57
KeyIn Strength Functions Shear/Normal 68
KeyIn Strength Functions Anisotropic 79
KeyIn Tension Crack 80
KeyIn Points 83
KeyIn Lines 85
KeyIn Slip Surface Grid & Radius 89
KeyIn Slip Surface Axis 93

KeyIn Slip Surface Specified 94
KeyIn Slip Surface Left Block 96
KeyIn Slip Surface Right Block 99
KeyIn Slip Surface Limits 101
KeyIn Pore Pressure: Water Pressure 102
KeyIn Pore Pressure: Air Pressure 109
KeyIn Load: Line Loads 110
KeyIn Load: Anchor Loads 111
KeyIn Load: Seismic Load 114
KeyIn Pressure Lines 115

The Draw Menu 118
Draw Points 119
Draw Points on Mesh 119
Draw Lines 120
Draw Slip Surface Grid 123
Draw Slip Surface Radius 126
Draw Slip Surface Axis 129
Draw Slip Surface Specified 130
Draw Slip Surface Left Block 132
Draw Slip Surface Right Block 137
Draw Slip Surface Limits 141
Draw Pore-Water Pressure 141
Draw Line Loads 146
Draw Anchor Loads 149
Draw Pressure Lines 152
Draw Tension Crack Line 155

The Sketch Menu 157
Sketch Lines 157
Sketch Circles 158
Sketch Arcs 158
Sketch Text 159
Sketch Axes 165

The Modify Menu 166
Modify Objects 166
Modify Text 169
Modify Pictures 171

The Tools Menu 176
Tools Verify 176
Tools SOLVE 179
Tools CONTOUR 179

The Help Menu 180

Introduction

The first step in a slope stability analysis is to define the problem. SLOPE/W DEFINE is an interactive and graphical function for accomplishing the definition part of an analysis.

This chapter describes the purpose, operation, and action of each SLOPE/W DEFINE command. The DEFINE commands are accessible by making selections from both the DEFINE menus and toolbars. The toolbars contain icons which invoke many of the commands available in the menus. The menus available and the function of each are as follows:

- ***File*** Opens, imports, and saves files and prints the drawing. For more information about this command, see The File Menu in this chapter.
- ***Edit*** Copies the drawing to the Clipboard. For more information about this command, see The Edit Menu in this chapter.
- ***Set*** Sets page, scale, grid, zoom and axes settings. For more information about this command, see The Set Menu in this chapter.
- ***View*** Controls viewing options and displays point and soil property information. For more information about this command, see The View Menu in this chapter.
- ***KeyIn*** Allows for typing in problem data. For more information about this command, see The KeyIn Menu in this chapter.
- ***Draw*** Defines problem data by drawing. For more information about this command, see The Draw Menu in this chapter.
- ***Sketch*** Defines graphic objects to label, enhance, and clarify the problem definition. For more information about this command, see The Sketch Menu in this chapter.
- ***Modify*** Allows graphic and text objects to be moved or deleted and text objects or pictures to be modified. For more information about this command, see The Modify Menu in this chapter.
- ***Tools*** Allows verification of problem data and gives quick access to running SOLVE and CONTOUR. For more information about this command, see The Tools Menu in this chapter.
- ***Help*** Displays the online help system and information about SLOPE/W. For more information about this command, see The Help Menu in this chapter.

In the remainder of this chapter, the commands in the toolbars and in each of these menus are presented and described.

Toolbars

Toolbars are small windows that contain buttons and controls to help perform common tasks quickly. Pressing a toolbar button is usually a shortcut for a command accessible from the menu; therefore, less time and effort is required to invoke a command from a toolbar than from a menu.

You can choose to display or hide toolbars. To toggle the display of a toolbar, use the View Toolbars command, or put the cursor on a displayed toolbar and click the right mouse button. For more information on the View Toolbars command, see View Toolbars in this chapter.

Toolbars are movable and dockable and may be reshaped. Movable means you can move a toolbar by dragging it with the mouse to any location on the display. Dockable means you can "dock" a toolbar at various locations on the display such as below the menu bar, or on the sides or bottom of the main window. You can reshape a toolbar by dragging the corner of the toolbar with the mouse. As this is done, the toolbar outline changes to reflect its new shape. The best way to get a feel for moving, docking and reshaping toolbars is to try these things yourself using the mouse.

In DEFINE, five toolbars are available for performing various tasks:

Standard Toolbar Contains buttons for file operations, printing, copying, redrawing and accessing other SLOPE/W programs. For more information about this toolbar, see Standard Toolbar in this chapter.

Mode Toolbar Contains buttons for entering different operating modes which are used to display and edit graphic and text object data. For more information about this toolbar, see Mode Toolbar in this chapter.

View Preferences Toolbar Contains buttons for toggling various display preferences. For more information about this toolbar, see View Preferences Toolbar in this chapter.

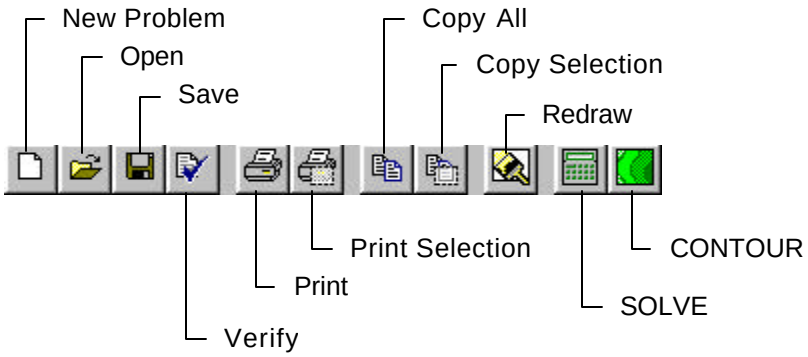
Grid Toolbar Contains controls for specifying the display of a drawing grid. For more information about this toolbar, see Grid Toolbar in this chapter.

Zoom Toolbar Contains controls for zooming in and out of the drawing. For more information about this toolbar, see Zoom Toolbar in this chapter.

Standard Toolbar

The Standard toolbar, shown in Figure 4.1, contains commands for initializing new problems, opening previously saved problems, saving a current problem, verifying the geometry, printing the current problem, copying the current problem to the Windows clipboard, redrawing the display, and starting the SOLVE and CONTOUR programs.

Figure 4.1 The Standard Toolbar



The toolbar buttons are:

New Problem Use the New Problem button to clear any existing problem definition data and reset DEFINE back to the user-defined default settings. This places DEFINE in the same state as when it was first invoked. This button is not the same as the File New command--for information about the File New command, see File New in this chapter.

Open Use the Open button as a shortcut for the File Open command. For information about this command, see File Open in this chapter.

Save Use the Save button as a shortcut for the File Save command. For information about this command, see The File Menu in this chapter.

Verify Use the Verify button as a shortcut for the Tools Verify command. For more information about this command, see Tools Verify in this chapter.

Print Use the Print button as a shortcut for the File Print command. For more information about this command, see File Print in this chapter.

Print Selection Use the Print Selection button to print a selected area of the drawing. For more information, see Print Selection Button below.

Copy All Use the Copy All button as a shortcut for the Edit Copy All command. For information about this command, see Edit Copy All in this chapter.

Copy Selection Use the Copy Selection button to copy a selected area of the drawing to the Windows Clipboard. For more information, see Copy Selection Button below.

Redraw Use the Redraw button as shortcut for the View Redraw command. For information about this command, see View Redraw in this chapter.

SOLVE Use the SOLVE button as a shortcut for the Tools SOLVE command. For information about this command, see Tools SOLVE in this chapter.

CONTOUR Use the CONTOUR button as a shortcut for the Tools CONTOUR command. For information about this command, see Tools CONTOUR in this chapter.

Print Selection Button

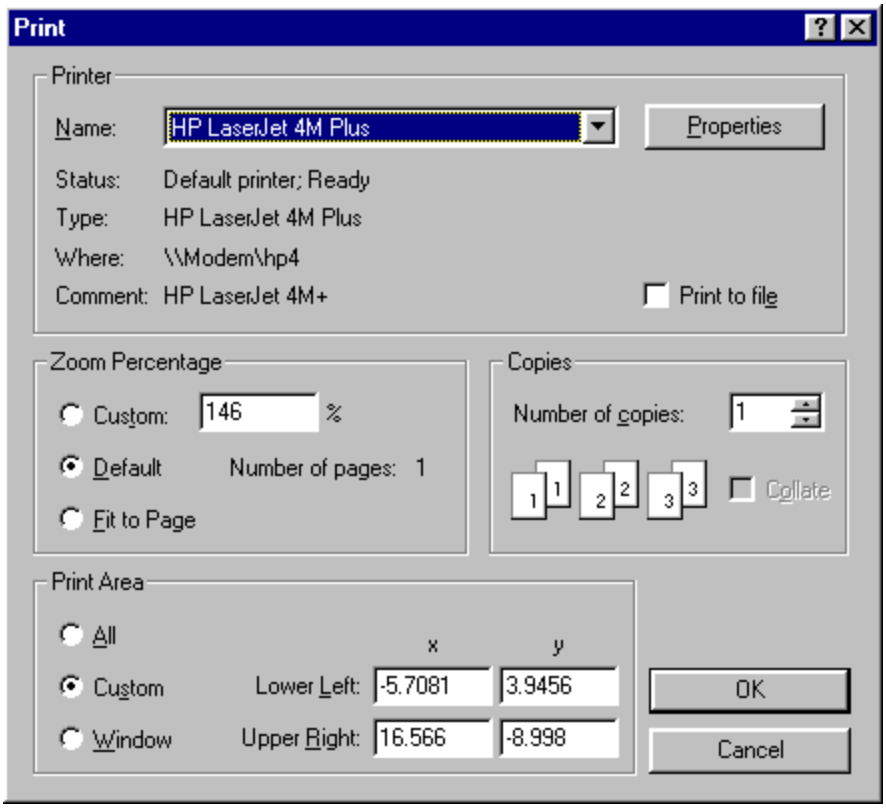
Use the Print Selection button to print a selected rectangular area of the drawing.

➢ **To print a selected area of the drawing:**

1. Press the Print Selection button on the Standard toolbar.

   The cursor changes to a cross-hair and the status bar indicates that “Print Selection” is the current mode.

2. The area of the drawing to copy is defined by dragging a rectangle around the region. For example, move the cursor to the top-left corner of the region. Push the left mouse button down, but do not release it. Now move the mouse to the right, and a rectangle appears. Drag the mouse until the rectangle encompasses the region to copy. The following dialog box appears:



For more information about the Print dialog box, see File Print in this chapter.

3. Click OK to send the selected area to the printer.

Copy Selection Button

Use the Copy Selection button to copy a selected area of the drawing to the Windows Clipboard. For information about the Windows Clipboard, see your Microsoft Windows documentation.

➢ **To copy a selected area of the drawing to the Clipboard:**

1. Click on the Copy Selection button with the left mouse button.

   The cursor changes to a cross-hair and the status bar indicates that "Copy Selection" is the current mode.

2. The area of the drawing to copy is defined by dragging a rectangle around the region. Move the cursor to the top-left corner of the region. Push the left mouse button down, but do not release it. Now move the mouse to the right, and a rectangle appears. Drag the mouse until the rectangle encompasses the region to copy.

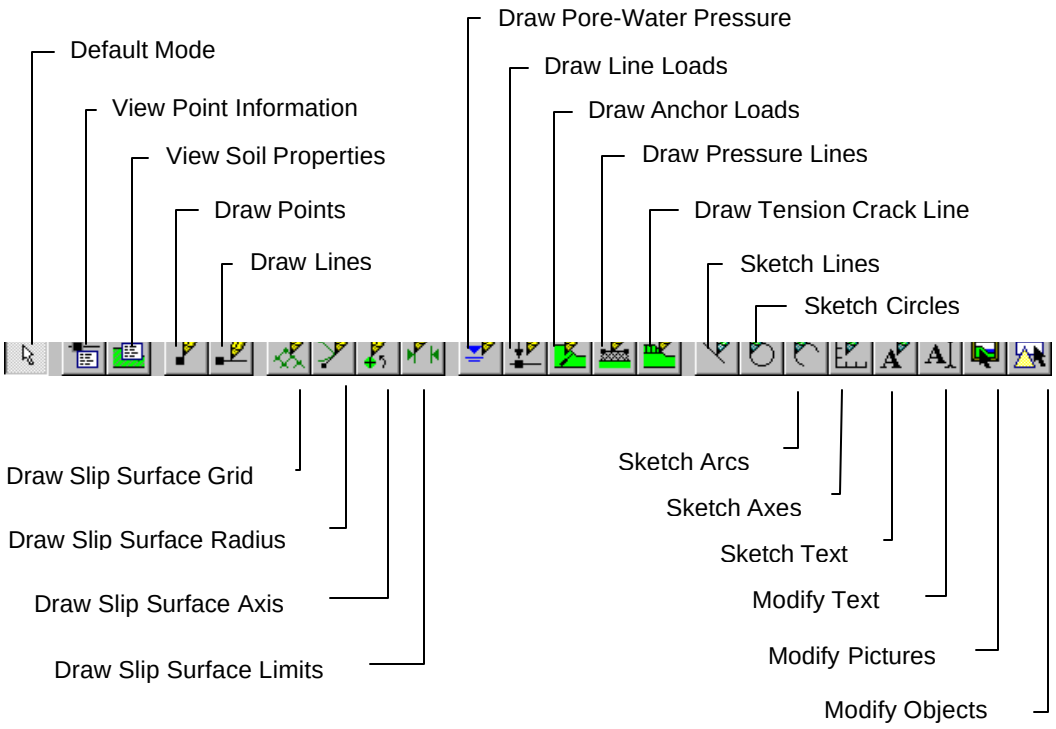
3. Release the left mouse button.

   A beep is sounded when the selected region has been copied to the clipboard. The Copy button returns to its normal state.

Mode Toolbar

The Mode toolbar, shown in Figure 4.2, contains buttons that put DEFINE into different "modes" used to accomplish specific tasks such as viewing point information and soil properties, drawing points and lines, drawing slip surface information for the current slip surface method (e.g., grid and radius lines, slip surface limits), drawing line and anchor loads, drawing a tension crack line, drawing pressure lines, drawing sketch objects and text, and modifying objects and pictures.

Figure 4.2 The Mode Toolbar



The toolbar buttons are:

Default Mode Use the Default Mode button to exit any current mode and return to the default mode.

View Point Information Use the View Point Information button as a shortcut for the View Point Information command. For more information about this command, see View Point Information in this chapter.

View Soil Properties Use the View Soil Properties button as a shortcut for the View Soil Properties command. For more information about this command, see View Soil Properties in this chapter.

Draw Points Use the Draw Points button as a shortcut for the Draw Points command. For more information about this command, see Draw Points in this chapter.

Draw Points on Mesh Use the Draw Points on Mesh button as a shortcut for the Draw Points on Mesh command. This button only appears on the Mode toolbar when a mesh has been imported and a finite element pore-water pressure option has been selected using KeyIn Analysis Control. For more information about this command, see Draw Points on Mesh in this chapter.

Draw Lines Use the Draw Lines button as a shortcut for the Draw Lines command. For more information about this command, see Draw Lines in this chapter.

Draw Slip Surface Grid Use the Draw Slip Surface Grid button as a shortcut for the Draw Slip Surface Grid command. This button only appears on the Mode toolbar when the Grid and Radius slip surface option has been selected using KeyIn Analysis Control. For more information about this command, see Draw Slip Surface: Grid in this chapter.

Draw Slip Surface Radius Use the Draw Slip Surface Radius button as a shortcut for the Draw Slip Surface Radius command. This button only appears on the Mode toolbar when the Grid and Radius slip surface option has been selected using KeyIn Analysis Control. For more information about this command, see Draw Slip Surface: Radius in this chapter.

Draw Slip Surface Axis Use the Draw Slip Surface Axis button as a shortcut for the Draw Slip Surface Axis command. For more information about this command, see Draw Slip Surface: Axis in this chapter.

Draw Slip Surface Specified Use the Draw Slip Surface Specified button as a shortcut for the Draw Slip Surface Specified command. This button only appears on the Mode toolbar when the Specified slip surface option has been selected using KeyIn Analysis Control. For more information about this command, see Draw Slip Surface: Specified in this chapter.

Draw Slip Surface Left Block Use the Draw Slip Surface Left Block button as a shortcut for the Draw Slip Surface Left Block command. This button only appears on the Mode toolbar when the Block slip surface option has been selected using KeyIn Analysis Control. For more information about this command, see Draw Slip Surface: Left Block in this chapter.

Draw Slip Surface Right Block Use the Draw Slip Surface Right Block button as a shortcut for the Draw Slip Surface Right Block command. This button only appears on the Mode toolbar when the Block slip surface option has been selected using KeyIn Analysis Control. For more information about this command, see Draw Slip Surface: Right Block in this chapter.

Draw Slip Surface Limits Use the Draw Slip Surface Limits button as a shortcut for the Draw Slip Surface Limits command. For more information about this command, see Draw Slip Surface: Limits in this chapter.

Draw Pore-Water Pressure Use the Draw Pore-Water Pressure button as a shortcut for the Draw Pore-Water Pressure command. This button only appears on the Mode toolbar when a non-finite element pore-water pressure option has been selected using KeyIn Analysis Control. For more information about this command, see Draw Pore-Water Pressure in this chapter.

Draw Line Loads Use the Draw Line Loads button as a shortcut for the Draw Line Loads command. This button will not appear on the Mode toolbar if the Finite Element Stress method has been selected using KeyIn Analysis Method. For more information about this command, see Draw Line Loads in this chapter.

Draw Anchor Loads Use the Draw Anchor Loads button as a shortcut for the Draw Anchor Loads command. This button will not appear on the Mode toolbar if the Finite Element Stress method has been selected using KeyIn Analysis Method. For more information about this command, see Draw Anchor Loads in this chapter.

Draw Pressure Lines Use the Draw Pressure Lines button as a shortcut for the Draw Pressure Lines command. This button will not appear on the Mode toolbar if the Finite Element Stress method has been selected using KeyIn Analysis Method. For more information about this command, see Draw Pressure Lines in this chapter.

Draw Tension Crack Line Use the Draw Tension Crack Line button as a shortcut for the Draw Tension Crack Line command. This button will only appear on the Mode toolbar if the Tension Crack Line option has been selected using KeyIn Analysis Control. For more information about this command, see Draw Tension Crack Line in this chapter.

Sketch Lines Use the Sketch Lines button as a shortcut for the Sketch Lines command. For more information about this command, see Sketch Lines in this chapter.

Sketch Circles Use the Sketch Circles button as a shortcut for the Sketch Circles command. For more information about this command, see Sketch Circles in this chapter.

Sketch Arcs Use the Sketch Arcs button as a shortcut for the Sketch Arcs command. For more information about this command, see Sketch Arcs in this chapter.

Sketch Axes Use the Sketch Axes button as a shortcut for the Sketch Axes command. For more information about this command, see Sketch Axes in this chapter.

Sketch Text Use the Sketch Text button as a shortcut for the Sketch Text command. For more information about this command, see Sketch Text in this chapter.

Modify Text Use the Modify Text button as a shortcut for the Modify Text command. For more information about this command, see Modify Text in this chapter.

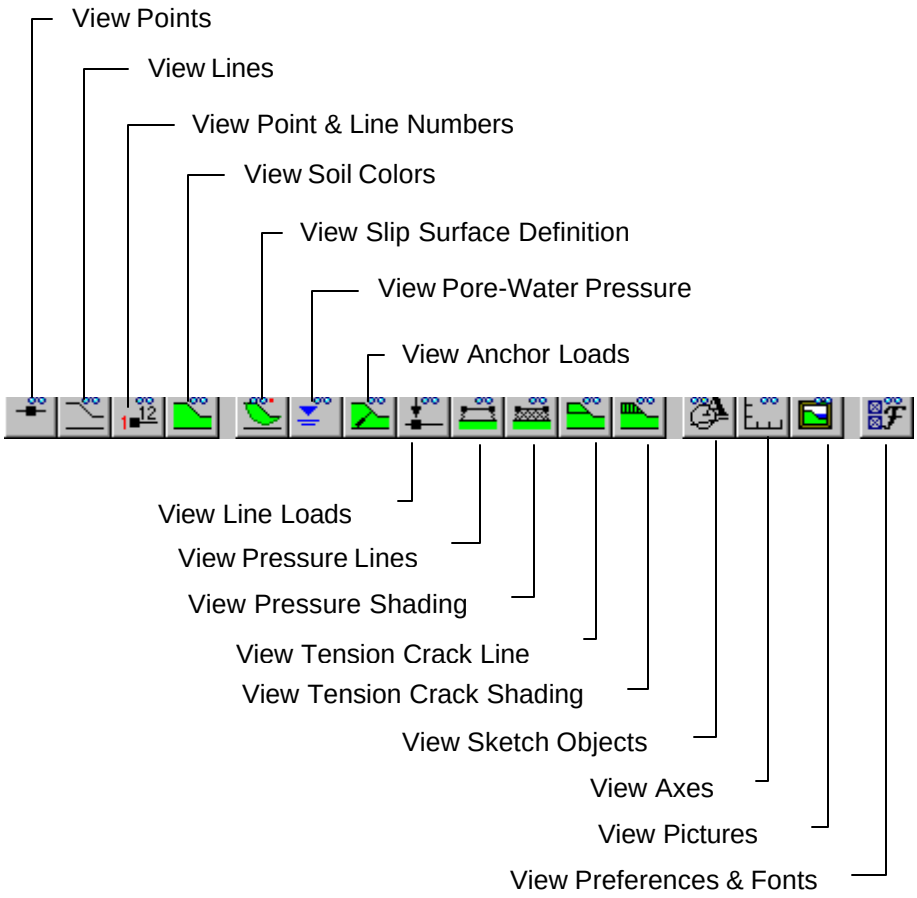
Modify Pictures Use the Modify Pictures button as a shortcut for the Modify Pictures command. For more information about this command, see Modify Pictures in this chapter.

Modify Objects Use the Modify Objects button as a shortcut for the Modify Objects command. For more information about this command, see Modify Objects in this chapter.

View Preferences Toolbar

The View Preferences toolbar, shown in Figure 4.3, contains buttons for setting viewing preferences such as points and lines and their numbers, soil colors, pore-water pressures, slip surface definitions, anchor and line loads, pressure lines and shading, sketch objects and text, pictures, text fonts, and the axes.

Figure 4.3 The View Preferences Toolbar



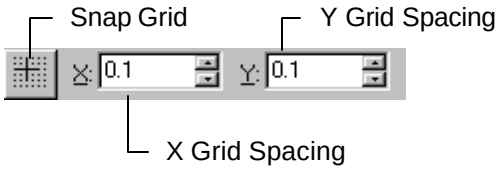
All the buttons on the View Preferences toolbar are shortcuts for the options accessible using the View Preferences command. For more information about this command, see View Preferences in this chapter.

NOTE: Some buttons will only appear on the View Preferences toolbar if the problem requires them. For example, a View Finite Element Mesh button appears only if a finite element mesh has been imported into the problem; the View Tension Crack Line and Shading buttons appear only if the tension crack line option has been selected using the KeyIn Analysis Control command.

Grid Toolbar

The Grid toolbar, shown in Figure 4.4, contains a button for toggling the display of grid points and controls for setting the *x* and *y* grid spacing.

Figure 4.4 The Grid Toolbar



The Grid toolbar allows you to quickly change your background grid spacing. For example, if you are drawing lines and you wish to refine the background grid, click on the down-arrow beside the X or Y grid spacing edit box; the grid spacing is reduced by half and the background grid is redrawn. You can then continue to draw lines.

The toolbar controls are:

Snap Grid Use the Snap Grid button as a shortcut for toggling both the grid display and snap to grid feature simultaneously.

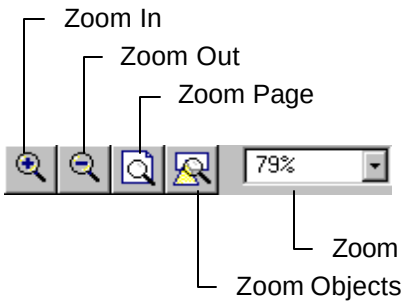
X and Y Grid Spacing Use the grid spacing controls to set the *x* and *y* grid spacing by either typing a value in the edit boxes or by using the spin controls adjacent to each edit box. Note that when the spacing value in one edit box is changed, the spacing value in the other edit box is automatically updated such that regular (i.e. square) grid will be generated on the display. Note also that if the drawing scale is different in the *x*- and *y*-directions, then the automatic updating of either the *x*- or *y*-spacing values will reflect this difference.

For more information on changing the background grid, see Set Grid in this chapter.

Zoom Toolbar

The Zoom toolbar, shown in Figure 4.5, contains buttons for zooming in and out of the drawing and a control for displaying and setting the zoom factor.

Figure 4.5 The Zoom Toolbar



The toolbar controls are:

Zoom In Use this button to zoom in on a user specified region. When the button is pressed, the cursor changes to a magnifying glass with a plus sign, ( ) and the status bar indicates that "Zoom In" is the current mode. You can then specify the region to be enlarged by using the mouse to drag a rectangle over the region. The display is then redrawn to show the region inside the specified rectangle.

Zoom Out Use this button to return to the previously viewed region. If there is no previously set region, then the full page is displayed.

Zoom Page Use this button to display the entire printable page.

Zoom Objects Use this button to display all defined objects in the window. The smallest region that encompasses all objects (i.e., points, soil lines, sketch objects, etc.) is calculated, and this region is displayed in the window.

Zoom Control This control shows the current size at which the drawing is displayed. When you push one of the other buttons on the Zoom toolbar, this control shows the new drawing display size. You also can use this control to specify any other display size. For example, to show the drawing at its specified scale, click on the down-arrow and select 100%; to show the drawing at 175%, type 175 in the Zoom control edit box and press the Enter key.

The Set Zoom command can also be used to change the drawing display size. For more information, see Set Zoom in this chapter.

The File Menu

The File menu commands are:

- ***New*** Initializes DEFINE for a new problem. For more information about this command, see File New in this chapter.
- ***Open*** Opens and reads an existing DEFINE data file. For more information about this command, see File Open in this chapter.
- ***Import: Data File*** Imports data from a PC-SLOPE, SEEP/W or SIGMA/W file. For more information about this command, see File Import: Data File in this chapter.
- ***Import: Picture*** Imports a bitmap or metafile into the current drawing. For more information about this command, see File Import: Picture in this chapter.
- ***Export*** Saves drawing in a format suitable for exporting to other programs. For more information about this command, see File Export in this chapter.
- ***Save*** Saves the current problem definition. File Save writes the current problem definition to the data file name displayed in the DEFINE window title bar. If the current problem definition is untitled, the File Save As dialog box appears.
- ***Save As*** Saves the current problem definition to an alternate data file. For more information about this command, see File Save As in this chapter.
- ***Save Default Settings*** Saves current settings as default settings. For more information about this command, see File Save Default Settings in this chapter.
- ***Print*** Prints the drawing. For more information about this command, see File Print in this chapter.
- ***Most Recently Used File*** Allows quick opening of one of the last six files opened. Selecting a file from the list is a convenient method for opening a recently used file.
- ***Exit*** File Exit quits DEFINE but does not quit Windows. You are prompted to save the current problem definition if any changes have been made.

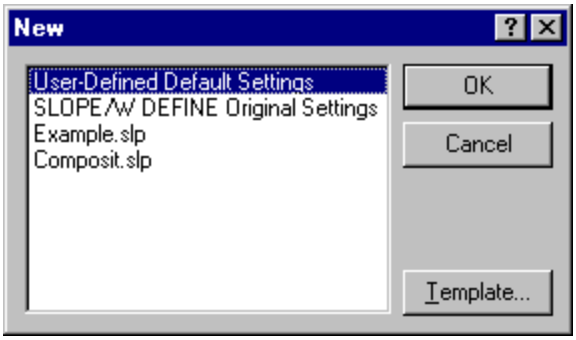
File New

☐ Initializes DEFINE for a new problem.

The File New command clears any existing problem definition data and initializes DEFINE for a new problem. You can initialize your new problem using DEFINE's default settings or the default settings that you have saved with the File Save Default Settings command. Alternatively, you can use an old problem as a template for your new problem; all soil properties, geometry, and other settings in the old problem will be used as a default "template" for your new problem.

➢ **To create a new problem:**

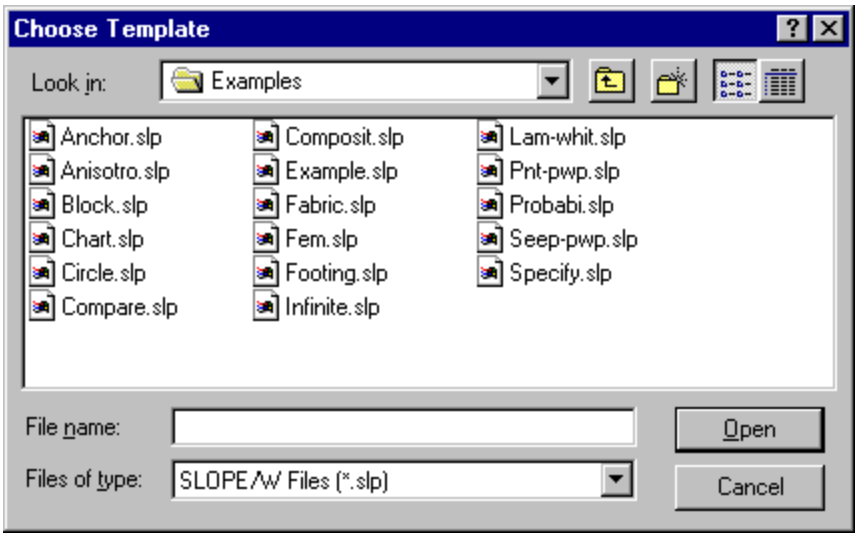
1. Choose New from the File menu. The following dialog box appears:



2. To create a new problem using the default settings that you have saved with File Save Default Settings, select User-Defined Default Settings in the list box.

3. To create a new problem using DEFINE's default settings, select SLOPE/W DEFINE Original Settings in the list box.

4. To create a new problem using an old problem as a template, select one of the filenames in the list box. If no file names are listed or if you wish to use a different file name as a template, select the Template button. The following dialog box appears:



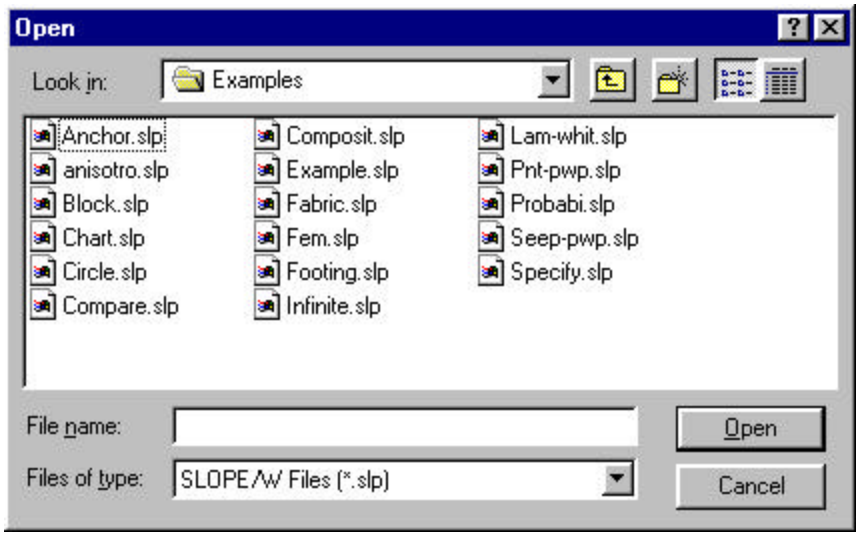
Select the file name to use as a template, and then select the Open button. The selected file name will be displayed in the File New list box.

5. Select OK in the File New dialog box to create the new problem based on the selected list box option.

File Open

☐ Opens and reads an existing DEFINE data file.

When you choose File Open, the following dialog box appears:



➢ **To open a file:**

- Type a name in the File Name edit box and then press OK. The file name may include a directory and a path. The file name extension must be omitted or entered as **SLP**.

  -- or --

- Click on a file name in the list box and then press OK.

  -- or --

- Double-click on a file name in the list box.

➢ **To change the current directory or drive:**

- Use the Look In box to select the drive and directory.

Use the other controls in the dialog box to navigate to the drive and directory containing the SLOPE/W file you wish to open.

NOTE: The SLOPE/W File Open dialog box is a common dialog used by many other Windows applications. To get help on using the dialog box, click on the question mark in the top-right corner; your cursor then becomes a question mark. Then, click on the dialog control that you need explained; a pop-up window appears with a description of the dialog control. Click anywhere else in the dialog box to remove the pop-up window.

Files Read by DEFINE

Two files are read when a DEFINE data file is opened. One has a file name extension of **SLP** and the other has an extension of **SL2**.

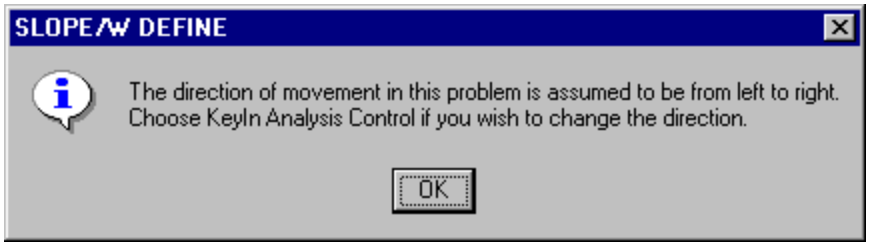
- The **SLP** file contains the data required for the slope stability calculations. It is also read by SOLVE and CONTOUR.

- The **SL2** file contains information relating to the graphical layout of the problem. (e.g. page size and units, engineering units and scale, sketch lines and text, and references to any imported picture files). It is also read by CONTOUR, but it is not required by SOLVE.

NOTE: When you open a problem containing imported picture files, SLOPE/W checks to see that the picture file names still exist. If a picture file has been moved or renamed, SLOPE/W displays the Import Picture dialog box, allowing you to specify a different picture file name in its place. See File Import: Picture or Modify Pictures for more information on importing pictures.

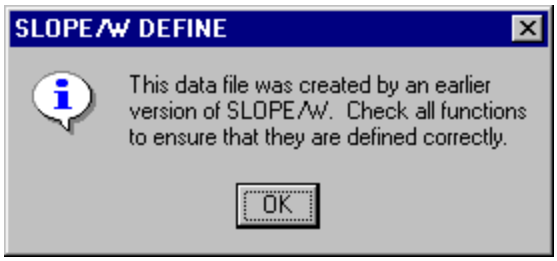
Reading Files Created by Earlier Versions of SLOPE/W

When you open a data file created by SLOPE/W Version 2, the following warning message is displayed:


SLOPE/W DEFINE
The direction of movement in this problem is assumed to be from left to right.
Choose KeyIn Analysis Control if you wish to change the direction.
OK

The direction of the slip surface movement is not directly specified in SLOPE/W Version 2. Therefore, SLOPE/W Version 3 and higher determines the movement direction by comparing the elevation of the Line 1 endpoints. If the left endpoint is higher, the movement direction is from left to right; if the right endpoint is higher, the movement direction is from right to left. While this will usually result in the correct movement direction being specified, in some cases, you may have to change the direction with the KeyIn Analysis Control command.

When you open a data file created by SLOPE/W Version 2 that uses strength functions, the following warning message is displayed:


SLOPE/W DEFINE
This data file was created by an earlier version of SLOPE/W. Check all functions to ensure that they are defined correctly.
OK

Since SLOPE/W Version 4 uses a more advanced spline interpolation technique than Version 2, you should use the KeyIn Functions commands to view all functions created in SLOPE/W Version 2. The spline curve passing through these data points may look different in SLOPE/W Version 4 than it did in SLOPE/W Version 2.

File Import: Data File

☐ Imports data from a PC-SLOPE, SEEP/W, or SIGMA/W file.

SLOPE/W is integrated with other GEO-SLOPE finite element Windows products. Using the File Import Data File command, you can import the node and element data from a finite element seepage or stress analysis. You can use the imported mesh as a guide to define the geometry for a corresponding slope stability analysis. SOLVE then can make use of the finite element-computed pore-water pressures and stresses to determine the stability.

You can also use the File Import Data File command to import a data file created by PC-SLOPE. PC-SLOPE is a text-based version of SLOPE/W for the MS-DOS operating system. All PC-SLOPE data files have a file name extension of **SET**.

➢ **To import a data file:**

The procedures for importing a file are the same as for the DEFINE File Open command. Specify the file name to import and select OK. The data file type to import can be selected from the Files of Type drop-down list box in the Import dialog box.

SLOPE/W will re-initialize your problem before importing the selected data file; you will be prompted to save your current problem, if necessary.

NOTE: The SLOPE/W File Import dialog box is a common dialog used by many other Windows applications. To get help on using the dialog box, click on the question-mark in the top-right corner; your cursor then becomes a question mark. Then, click on the dialog control that you need explained; a pop-up window appears with a description of the dialog control. Click anywhere else in the dialog box to remove the pop-up window.

Importing PC-SLOPE Files

After you have imported a PC-SLOPE file, you should check that all the parameters have been defined correctly, particularly the soil properties, analysis methods, and pore-water pressure method. As well, choose Tools Verify so that SLOPE/W can verify the problem data.

When a PC-SLOPE data file is imported, SLOPE/W sets the engineering scale and page origin so that the geometry will fit onto the default page size (defined in the Settings file). To select the appropriate engineering units, SLOPE/W compares the unit weight of water value with the default values for the various units. Choose Set Scale and Set Page if you wish to modify any of these parameters.

Importing Finite Element Files

You must import a finite element mesh if you want to use finite element computed pore-water pressures or stresses in the slope stability analysis.

You can define points directly on top of the finite element mesh nodes by choosing Draw Points on Mesh.

Two files are read when a SEEP/W or SIGMA/W data file is imported. One has a file name extension of **SEP** or **SIG** and the other has an extension of **SE2** or **SI2**. SLOPE/W reads the finite element mesh data from the **SEP** or **SIG** file and the page size, engineering scale, and units from the **SE2** or **SI2** file.

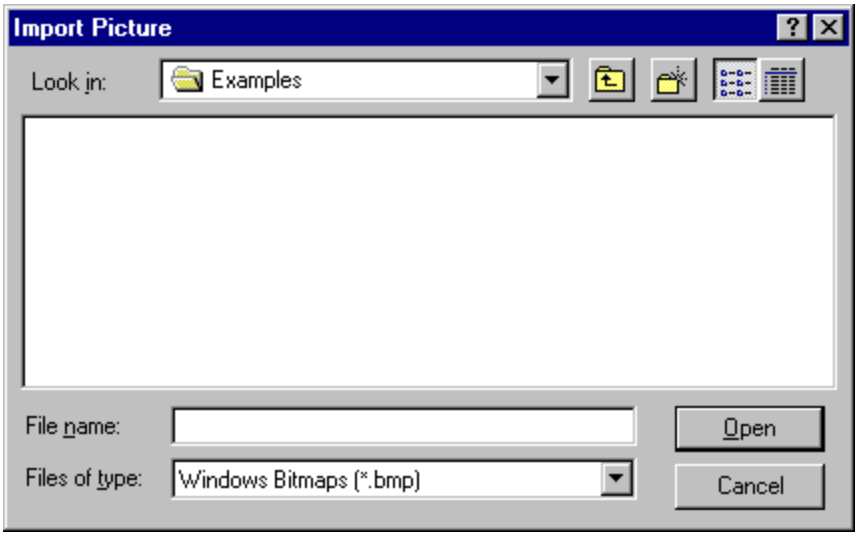
File Import: Picture

☐ Imports a bitmap or metafile into the current drawing.

File Import Picture allows you to place a bitmap or metafile picture on your drawing. For example, if you have a cross-section already defined in another Windows CAD or drawing program, you can save it as a WMF or EMF metafile, import it into SLOPE/W, and use your previously-defined cross-section as a background for drawing your SLOPE/W geometry. You also can use the File Import Picture command for inserting a company logo, photograph, or any other image into your SLOPE/W drawing.

➢ **To import a picture into the drawing:**

1. Choose Import: Picture from the File menu. The following dialog box appears:



NOTE: The SLOPE/W Import Picture dialog box is a common dialog used by many other Windows applications. To get help on using the dialog box, click on the question-mark in the top-right corner; your cursor then becomes a question mark. Then, click on the dialog control that you need explained; a pop-up window appears with a description of the dialog control. Click anywhere else in the dialog box to remove the pop-up window.

2. In the Files of Type drop-down list box, select the file format of the picture to import. You can import Windows bitmaps (.BMP), Windows 3.1 metafiles (.WMF), or Windows 95/NT metafiles (.EMF).

3. Specify the file name to import and select Open.

   The Import Picture dialog box disappears, the cursor changes from an arrow to a cross-hair, and the status bar indicates that "Import Picture" is the current operating mode.

4. Move the cursor to the position on the drawing where you wish to place the imported picture, and click the left mouse button.

   The picture is placed on the drawing such that the bottom-left corner is aligned with the cursor position.

5. Choose the Modify Objects command if you wish to change the size or position of the imported picture.

6. Choose the Modify Pictures command if you wish to change the picture ordering, to remove the picture, to change the file name that the picture is referenced to, or to scale the picture to match the current engineering scale.

NOTE: When you save your problem, SLOPE/W stores the file name that the picture is referenced to, rather than a copy of the imported picture. Therefore, if you later move or rename the picture file that you have just imported, you will have to re-establish the link to the new picture file the next time you open the problem in SLOPE/W.

Comments

You can import Windows bitmaps (.BMP), Windows 3.1 metafiles (.WMF), or Windows 95/NT metafiles (.EMF) into your drawing. The WMF format was developed for use by Windows 3.1 programs, but it is still used by some Windows 95 programs. The EMF format is an improved metafile format that retains more information about the

drawing; it is only used by 32-bit Windows applications. If you have a choice between importing a WMF metafile and an EMF metafile, you should import the EMF metafile.

To transfer your current SLOPE/W drawing into other Windows applications, see the File Export or Edit Copy All commands.

NOTE: The File Import Picture command *cannot* be used to import problem geometry from another GEO-SLOPE application. Imported picture (i.e., WMF and EMF) files do not contain any soil line or property information; they are only useful for display purposes. To import a mesh from a GEO-SLOPE finite element application, choose the File Import: Data File command.

File Export

☐ Saves drawing in a format suitable for exporting to other programs.

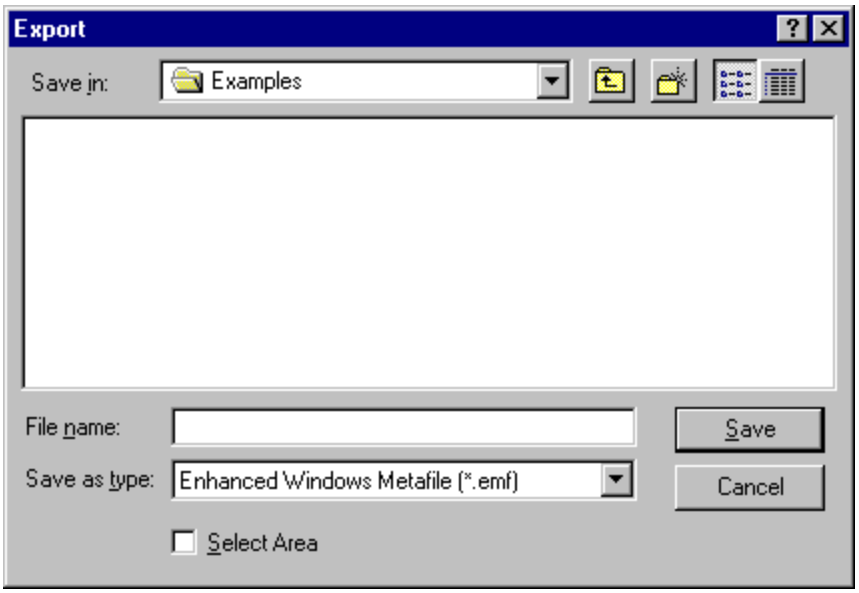
File Export saves your drawing in a format that can be read by other programs. This feature allows you to include your drawing in reports and presentations and to enhance your drawing using other drawing or CAD software packages.

The drawing can be exported in the Windows Metafile (WMF) format or the Enhanced Metafile (EMF) format. The WMF format was developed for use by Windows 3.1 programs, but it is still used by some Windows 95 programs. The EMF format is an improved metafile format that retains more information about the drawing; it is only used by 32-bit Windows applications.

Both the WMF and EMF file formats contain a graphical representation of your drawing only; SLOPE/W information (e.g., points, lines and soil properties) is not stored in either metafile format.

➢ **To export the drawing:**

1. Choose Export from the File menu. The following dialog box appears:



2. In the Save as Type drop-down list box, select the file format in which to save the drawing.

3. If you wish to select a region of the drawing to export, check the Select Area check box.

4. Type the name you wish to give the exported file, including extension, and select the directory in which to save the file.

5. Click OK. If the file name already exists, you may elect to over-write the existing file.

   If the Select Area check box is checked, then the cursor changes from an arrow to a cross-hair and the status bar indicates that the “Select Export Area” is the current mode; the area can now be selected.

   If the Select Area check box is cleared, then the entire drawing is exported to the specified file and a beep is sounded when the file export operation is completed.

6. The area of the drawing to export is defined by dragging a rectangle around the area. Move the cursor to the top-left corner of the area. Push the left mouse button down, but do not release it. Now move the mouse to the right, and a rectangle appears. "Drag" the mouse until the rectangle encompasses the area to export.

7. Release the left mouse button.

   A beep is sounded when the file export operation is completed.

Comments

The File Export, Edit Copy All, and Copy Selection Button commands can *all* be used to transfer your drawing to another application. The command you use will depend on the import capabilities of the other Windows application.

If you have imported any metafile pictures (using the File Import: Picture command), you should export your drawing using the EMF format, not the WMF format. Since the WMF format is incapable of storing embedded metafile pictures, you will not be able to see your imported pictures in an exported WMF file.

NOTE: The File Export command *cannot* be used to transfer your problem data into another SLOPE/W problem. WMF and EMF files do not contain any soil line or property information - only drawing primitives such as rectangles and lines. See the File New command for information on creating a new SLOPE/W problem based on a previously-defined problem.

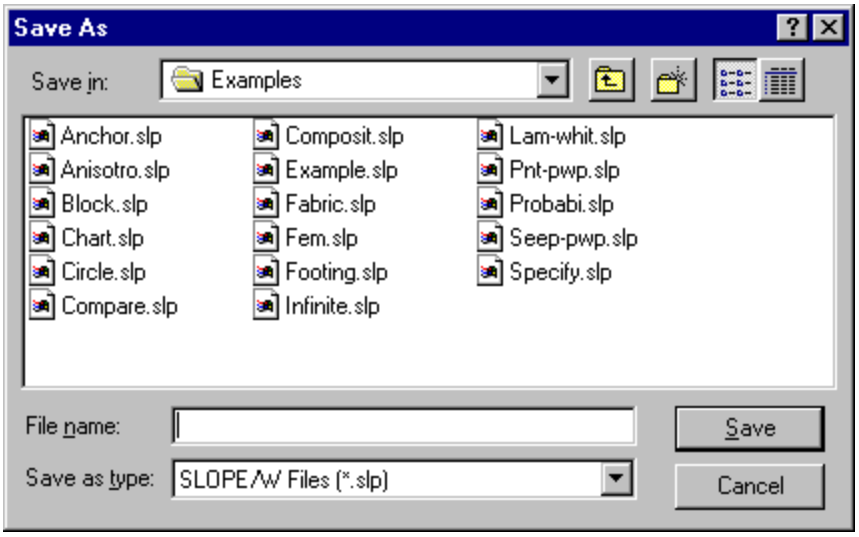
File Save As

☐ Saves the current problem definition to an alternate data file.

File Save As allows you to save the problem definition to an alternate file if you do not wish to modify the current file. The file name extension must either be omitted or must be **SLP**.

➢ **To save the drawing to an alternate data file:**

1. Choose Save As from the File menu. The following dialog box appears:



2. Type the name you wish to give the file and select the directory in which to save the file.

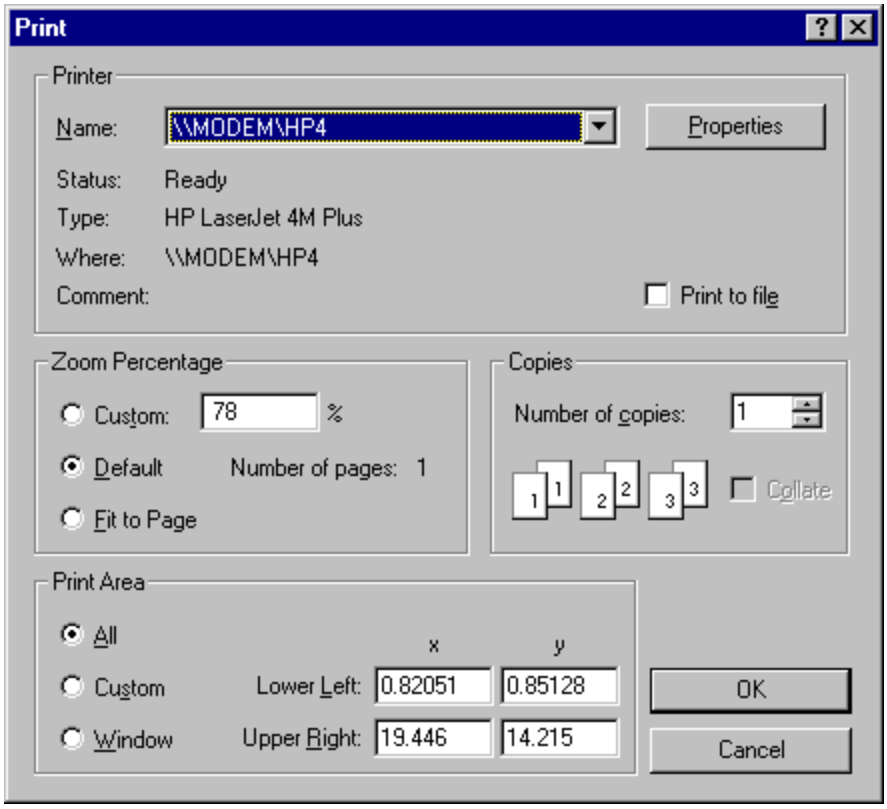
3. Select OK. If the file name already exists, you may elect to over-write the existing file.

NOTE: The SLOPE/W File Save As dialog box is a common dialog used by many other Windows applications. To get help on using the dialog box, click on the question-mark in the top-right corner; your cursor then becomes a question mark. Then, click on the dialog control that you need explained; a pop-up window appears with a description of the dialog control. Click anywhere else in the dialog box to remove the pop-up window.

File Print

☐ Prints the drawing.

When you choose File Print, the following dialog box appears:



Printer The printer group box contains controls for selecting the printer and changing its properties. Use the Name combo box to select the printer and use the Properties button to set printer settings. Check the Print to File checkbox if you wish to sent the print job to a file for printing later. For more information about printer settings, see your Windows documentation.

Zoom Percentage This group box defines the size at which to print the drawing and displays the number of pages required for printing. The size can be set to any percentage. The default size is equal to the currently displayed drawing size. When the Default button is pressed, the size is set to the default value. When the Fit to Page button is pressed, the size is changed so that the drawing will fill one entire printed page.

Print Area This is the area of the drawing that you wish to print. The edit boxes define the lower-left and upper-right corners of the rectangular area to print. When you select All to print the entire drawing, the coordinates of the lower-left and upper-right corners of the drawing are copied into the edit boxes. When you select Window to print only the portion of the drawing being displayed in the DEFINE window, the coordinates of the corners of the window are copied into the edit boxes.

➢ **To print the drawing:**

1. Specify the area of the drawing to print in the Print Area group box.
   - To print the entire drawing, select the All button.
   - To print only the portion of the drawing being displayed in the DEFINE window, select the Window button.

- To print any other rectangular portion of the drawing, type the coordinates of the lower-left and upper-right corners of the region in the edit boxes. The Custom button is selected.

When the area to print is selected, the Print Information group box is updated with the number of pages required to print.

2. Specify the size at which to print. Press the Fit to Page button if the area to print is to be fit on one page. Otherwise, the area to print will be printed at the specified size on as many pages as necessary.

   When the Fit to Page button is pressed, the value in the Custom edit box is changed so that the drawing will fill one entire printed page, and the number of pages printed is set to 1.

3. Select OK.

   DEFINE begins to send the drawing to the printer.

4. Select Cancel if you wish to abort the printing.

Comments

You can print the drawing at the exact engineering scale by printing at a size of 100%.

Printing jobs can be canceled from Windows. For more information on canceling print jobs, see your Windows documentation.

Only the objects currently displayed on the drawing are printed.

The drawing is printed in the center of the printer page.

The quickest way to specify a region to print is to select the Print Selection button from the Standard toolbar and drag a rectangle over the desired region. Typing the region coordinates in the Print Area edit boxes is useful if you already know the coordinate values.

Changing printer settings can help to resolve printing problems. For example, HP LaserJet 4 Series printers may not print rotated TrueType fonts at the correct angle or position. This problem can be overcome by sending the TrueType fonts directly to the printer instead of allowing the printer to rasterize the fonts. In the Printer Setup dialog box, select the Options button, change the Graphics Mode to Raster, and send the TrueType fonts as graphics.

File Save Default Settings

☐ Saves the current settings as the defaults.

This command allows you to save your current settings so that they can be used again when you define new problems. When you choose this command, the following settings are stored in the Windows registry:

- Working page units
- Engineering units
- View preferences
- Axis size and options
- Grid spacing and options
- Default colors used when specifying soil colors

When you choose the File New command, you can initialize the new problem with your default settings or with DEFINE's built-in default settings. Alternatively, you can use an old problem as a template for your new problem; all soil properties, geometry, and other settings in the old problem will be used as a default "template" for your new problem. For more information about initializing new problems, see File New in this chapter.

NOTE: When you open a problem using File Open, the default settings are replaced by the settings stored in the problem data files.

The Edit Menu

The Edit menu commands are:

- ***Copy All*** Copies the entire drawing to the Windows Clipboard. For more information about this command, see Edit Copy All in this chapter.

Edit Copy All

☐ Copies the entire drawing to the Windows Clipboard.

The Windows Clipboard provides temporary storage for information that you want to transfer between applications. The Edit Copy All command copies the entire drawing to the Clipboard for pasting into other applications. This is useful for preparing reports, slide presentations, or for adding further enhancements to the drawing. See your Windows documentation for further information on the Clipboard.

To copy the entire drawing to the Clipboard, choose the Copy All command from either the Edit menu or from the Standard toolbar. A beep is sounded when the drawing has been copied to the Clipboard.

Comments

To display the contents of the Clipboard, run the Clipboard Viewer program from Windows. For more information on the Clipboard Viewer, see your Windows documentation.

The Edit Copy All, Copy Selection Button button and the File Export command can all be used to transfer your drawing to another Windows application. The command you use will depend on the import capabilities of the other application.

You can also enhance your drawing by importing pictures into your drawing, rather than exporting your drawing to another Windows application for enhancement. See the File Import: Picture command for more information.

NOTE: The Edit Copy All command *cannot* be used to transfer your problem data into another SLOPE/W problem. The Clipboard memory does not contain any soil line or property information - only drawing primitives such as rectangles and lines. See the File New command for information on creating a new SLOPE/W problem based on a previously-defined problem.

The Set Menu

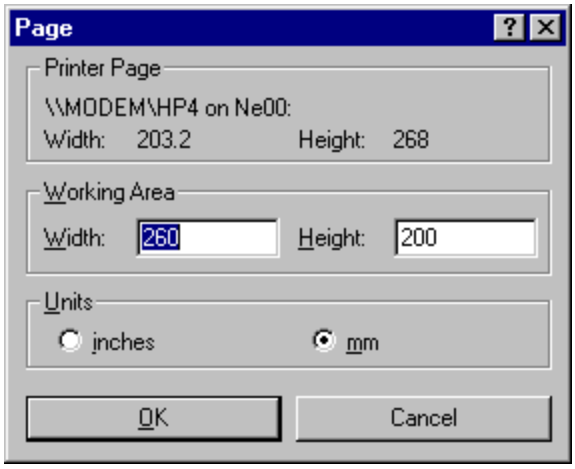
The Set menu commands are:

- ***Page*** Sets the size of the working area. For more information about this command, see Set Page in this chapter.
- ***Scale*** Sets the engineering scale, units, and unit weight of water. For more information about this command, see Set Scale in this chapter.
- ***Grid*** Creates a grid of points to assist in drawing objects. For more information about this command, see Set Grid in this chapter.
- ***Zoom*** Increases or decreases the size at which the drawing is displayed. For more information about this command, see Set Zoom in this chapter.
- ***Axes*** Defines scaled reference lines. For more information about this command, see Set Axes in this chapter.

Set Page

☐ Sets the size of the working area.

When you choose Set Page, the following dialog box appears:



Printer Page This group box displays the paper size used by the installed printer device. The paper size depends on the printer driver installed and on the printer setup configuration (see File Print to change the printer settings). These dimensions are displayed in the Printer Page group window to provide a guide for setting the working area.

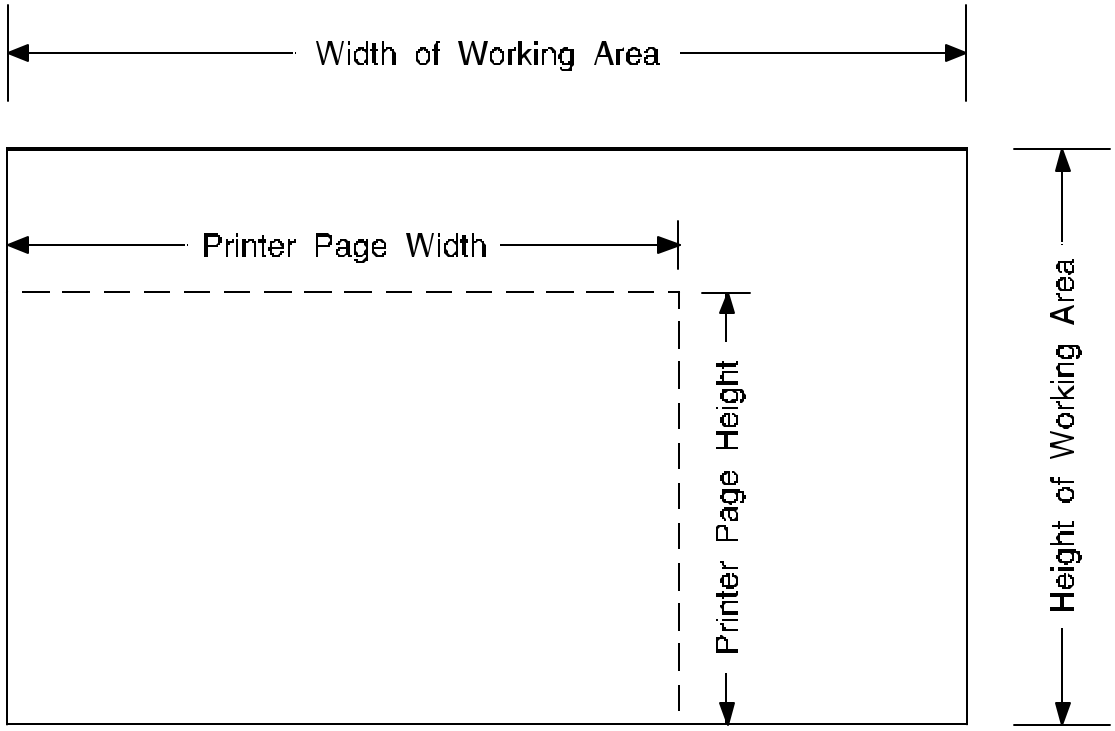
Working Area The working area represents the page size available for defining a problem. The printer page size is the size of a drawing that can be printed on one page with the installed printing device. If the working page is larger in height or width than the printer page, more than one sheet of paper is required to print the drawing at 100%. However, the drawing can be printed at a smaller size in order to fit on one page.

➢ **To set the working area size:**

1. Select the desired page units.
2. Type the desired width and height in the Width and Height edit boxes.

Figure 4.6 shows the relationship between the printer page and the working area.

Figure 4.6 Definition of Working Area and Printer Page



Comments

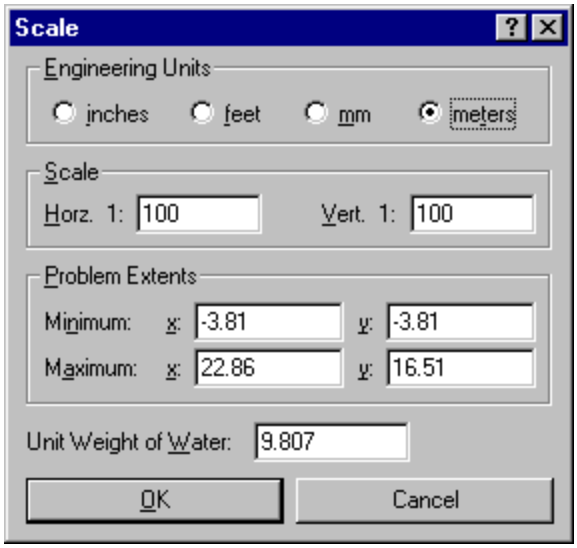
Choose the Zoom Page button to view the entire working area in the DEFINE window.

You should select a working area that allows you to work at a convenient engineering scale. This means that often your working area will need to be larger than the printer page.

Set Scale

☐ Sets the engineering scale, units, and unit weight of water.

When you choose Set Scale, the following dialog box appears:



Engineering Units The engineering units are the units used to measure the physical dimensions of the problem in the field.

Scale The scale is a ratio of the distance on a drawing to the actual physical distance in the field. For example, a 1:100 scale means that 1 unit on paper represents 100 units in the field. It could mean that 1 foot equals 100 feet or 1 meter equals 100 meters. Horz. 1: accepts the ratio of the horizontal drawing dimensions to the horizontal physical dimensions, and Vert. 1: accepts the ratio of the vertical drawing dimensions to the vertical physical dimensions. The scale ratio is not affected by the engineering units selected.

When the scale is changed, the problem extents are also changed to reflect the new engineering dimensions.

Problem Extents The problem extents define the engineering dimensions of your problem. All soil lines and other problem data must be contained within the problem extents. The problem extents are increased whenever you increase the scale or the size of the working area.

As you enter new values for the problem extents, the engineering scale is adjusted. This allows you to find an appropriate scale for the selected working area. When you enter the boundaries of your engineering problem in the Problem Extents edit boxes, the scale will be adjusted automatically. You can then adjust the scale to an even number of units. If the scale is too small, you may have to increase the size of the working area with the Set Page command.

NOTE: Do not specify the minimum problem extents as large values. Using a large starting *x*- or *y*-coordinate may affect the precision of the computed results due to round-off error. For example, it is better to specify the *y*-extents from 0 to 20 instead of from 7000 to 7020. For more information about round-off error, see Selecting Appropriate X and Y Coordinates in Chapter 7.

- **To set the scale if the engineering scale is known:**

1. Select the engineering units.
2. Type the minimum engineering coordinates in the Minimum-x and Minimum-y edit boxes.

   The Horz. and Vert. Scale values change to reflect the new engineering dimensions of the problem.
3. Type the scale ratio in the Horz. 1: and Vert. 1: edit boxes.

   The Maximum-x and Maximum-y values change to reflect the new engineering scale.
4. Select OK.

- **To set the scale if the extents are known:**

1. Select the engineering units.
2. Type the minimum engineering coordinates in the Minimum-x and Minimum-y edit boxes.

   The Horz. and Vert. Scale values change to reflect the new engineering dimensions of the page.
3. Type the maximum engineering coordinates in the Maximum-x and Maximum-y edit boxes.

   The Horz. and Vert. Scale values change to reflect the new engineering dimensions of the page.
4. If necessary, adjust the scale ratios in the Horz. 1: and Vert. 1: edit boxes to be in even units (e.g., if the Horz. Scale is 1:201.92 and the Vert. Scale is 1:214.27, you might set both scale ratios to be 1:200).

   The Maximum-x and Maximum-y values change to reflect the new engineering scale.
5. Select OK.

Unit Weight of Water The Unit Weight of Water must be specified for the purpose of converting pressure into head and vice versa. The units must be consistent with the units you selected for pressure and length. Table 4.1 gives examples and default values.

Table 4.1 Default Values for Unit Weight of Water

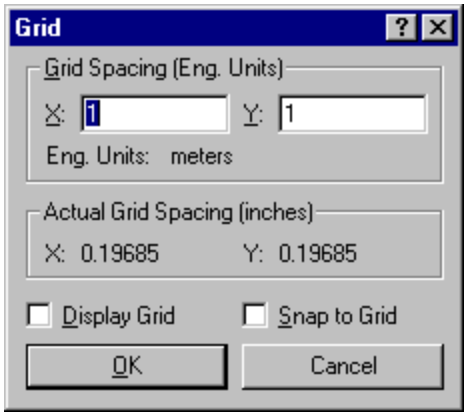
Soil Weight Unit	Length Unit	Cohesion Unit	Water Weight Unit	Default Unit Weight of Water
kN/m^3	m	kN/m^2 (kPa)	kN/m^3	9.807
N/mm^3	mm	N/mm^2	N/mm^3	9.807×10^{-6}
lbs/ft^3	feet	lbs/ft^2 (psf)	lbs/ft^3	62.4
lbs/in^3	inches	lbs/in^2 (psi)	lbs/in^3	0.03611

The default value is placed in the Unit Weight of Water edit box when you select the engineering units. This value may be changed by typing the appropriate value in the Unit Weight of Water edit box.

Set Grid

☐ Creates a grid of points to assist in drawing objects.

When you choose Set Grid, the following dialog box appears:



The grid is a pattern of dots which can be displayed to assist you in drawing objects (e.g. points, soil lines, text, etc.). When drawing an object, you can "snap" the object to the nearest grid point. This enables you to draw objects at precise coordinates.

➢ **To display and snap to the grid:**

1. Check the Display Grid check box.
2. Check the Snap to Grid check box.
3. Type the grid spacing in engineering units in the X and Y edit boxes.
4. Select OK.

Grid Spacing (Eng. Units) The X and Y values represent the distance between each grid point in the horizontal and vertical directions respectively. When a value is entered, the other value is recalculated so that the grid is evenly spaced.

Actual Grid Spacing Displays the actual distance between each grid point in the DEFINE window. This assists you in selecting an appropriate grid spacing in engineering units. This distance is displayed in either millimeters or inches, depending on which system of units was chosen for the working page size.

Display Grid Turns on and off the display of the grid on the drawing.

Snap to Grid Turns on and off the capability to snap to the grid when defining objects.

NOTE: Once you have used Set Grid to define your background grid, you will probably find the Grid Toolbar to be a more convenient way of modifying the grid spacing and turning the grid on and off.

Comments

To quickly enable or disable snapping to the background grid, click on the Snap Grid button in the Grid toolbar instead of choosing Set Grid. The Set Grid command is primarily used to change the spacing of the background grid.

DEFINE will always display the grid when Snap To Grid is on. Snap To Grid cannot be on if Display Grid is off.

If grid snapping is on, the cursor position displayed in the status bar reflects the position of the nearest grid point, not the actual cursor position. This allows you to see the position the cursor will snap to when you are drawing objects.

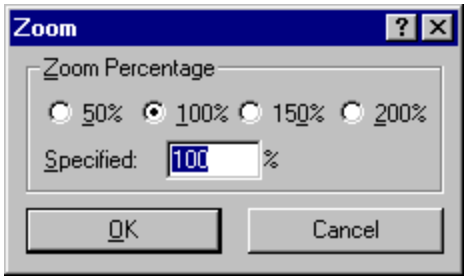
Displaying the grid may require significant computing and drawing time when the points are closely spaced. You can reduce the drawing display time by turning off the grid.

If the actual grid spacing is too small, the grid points will not be displayed. However, DEFINE will still snap to the grid when you draw objects.

Set Zoom

☐ Increases or decreases the size at which the drawing is displayed.

When you choose Set Zoom, the following dialog box appears:



Choosing Set Zoom allows you to increase or decrease the size at which the drawing is displayed and printed. Clicking on 100% displays the drawing at its original size; clicking on a different percentage changes the drawing size to the specified percentage. The drawing can be displayed at any size by typing the desired percentage in the Specified edit box.

The percentage must be a positive number greater than zero. The maximum percentage allowed is a function of the working page size, units, and scale; also, Windows NT allows you to specify a much larger zoom percentage than Windows 95. If you specify a zoom percentage that is too large, an error message will appear.

Comments

The simplest way to change the drawing display size is to use the Zoom toolbar. You may wish to use the Set Zoom command if the Zoom toolbar is not displayed.

Point symbols are limited in size to 200%. For example, when the rest of the drawing is displayed at 500%, point symbols are displayed at 200%. This feature makes it possible to see the points when they overlap at smaller sizes.

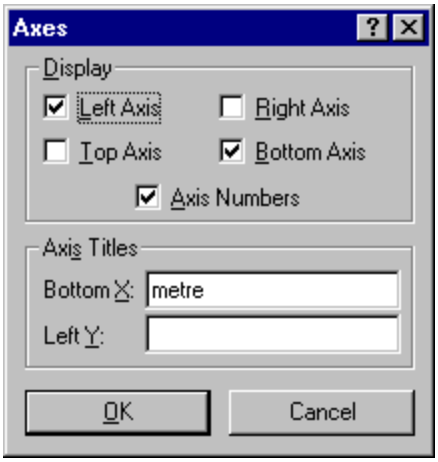
Set Axes

☐ Defines scaled reference lines.

Scaled and labeled reference axes can be generated at any suitable place on the drawing.

➢ **To generate reference axes:**

1. Choose Axes from the Set menu. The following dialog box appears:

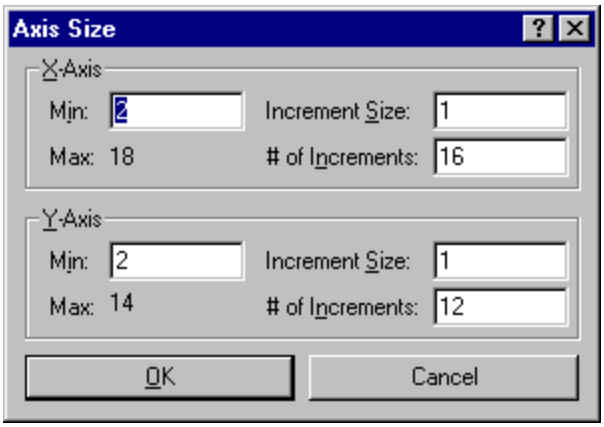


2. In the Display group box, check the sides of the axis you wish to display. For example, if you check Left Axis, Right Axis, Top Axis, and Bottom Axis, a rectangular axis is generated with tick marks on all four sides. Any combination of the four axes may be checked. Axes which are unchecked will not be drawn.

3. Check the Axis Numbers check box if you wish to number the axis tick marks.

4. Type a suitable title for the bottom and left sides of the axes in the Bottom X and Left Y edit boxes, respectively.

5. Select OK. The Axes dialog box appears.



6. Type the appropriate values in the X-Axis and Y-Axis group boxes.

   - ***Min*** Contains the minimum value displayed on the axis.
   - ***Increment Size*** Controls the spacing of the tick marks along the axis.
   - ***# of Increments*** Controls the length of the axis.
   - ***Max*** This is the highest value on the axis. It is displayed to provide a guide to selecting the increment size and number of increments along the axis.

7. Select OK. An axis is generated on the drawing.

NOTE: Axes can be moved and resized with the Modify Objects command.

Comments

Only one set of axes can be defined on a drawing.

The View Preferences command allows you to change the font and the size of the axes numbers and labels.

The axes can also be generated with the Sketch Axes command. You may find it convenient to first sketch the axes at an approximate location and size, and then choose Set Axes to refine the controlling parameters. Alternatively, you can move and resize the axes with the Modify Objects command once the axes are defined.

The View Menu

The View menu commands are:

- ***Point Information*** Displays information about the selected point. For more information about this command, see View Point Information in this chapter.
- ***Soil Properties*** Displays information about the selected soil or soil line. For more information about this command, see View Soil Properties in this chapter.
- ***Preferences*** Identifies which items will be displayed on the drawing. For more information about this command, see View Preferences in this chapter.
- ***Toolbars*** Displays or hides the DEFINE toolbars and the status bar. For more information about this command, see View Toolbars in this chapter.
- ***Redraw*** Redraws the problem. For more information about this command, see View Redraw in this chapter.

View Point Information

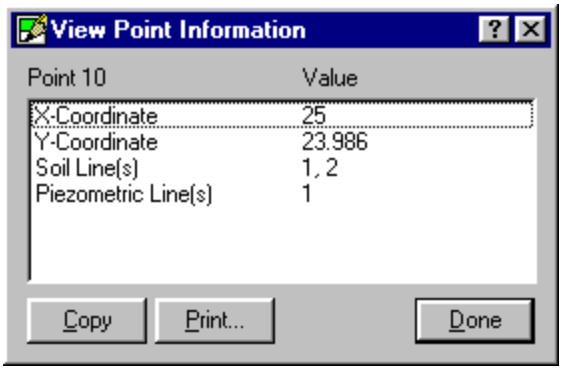
☐ Displays information about the selected point.

➢ **To view point information:**

1. Choose the View Point Information command from either the DEFINE menu or from the Mode toolbar.

   The cursor changes from an arrow to a cross-hair, and the status bar indicates that "View Point Information" is the current mode.

2. Move the cursor near the desired point and click the left mouse button. The point is selected and the following dialog box is displayed, containing the point information:



   The dialog box lists the *x*- and *y*-coordinates of the point, any lines in which the point may be contained, and any line load or anchor load values at the point. The dialog box also indicates if the point is part of the slip surface grid, radius, or block.

3. To see all the soil properties, re-size the dialog box by dragging the bottom edge of the window down until all information is displayed.

4. Repeat Step 2 for every point that you wish to view.

5. To copy the point information to the Windows Clipboard, select Copy. The point information is copied to the Clipboard in the following text format:

```
Point 3
X-Coordinate              20
Y-Coordinate              9
Line Load Magnitude       10
Line Load Direction       45
Soil Line(s)              1, 2
Piezometric Line(s)       1
```

6. To print the point information on the current printer, select Print. The point information is printed in the same format as for copying to the Clipboard.

7. Select Done, press ESC, or click the right mouse button to finish viewing point information.

Comments

To manually edit the point coordinates, choose KeyIn Points. To delete or move points, choose Modify Objects. Other information defined at the point can be changed using the corresponding KeyIn or Draw command.

See the View Soil Properties command for information on displaying soil properties.

View Soil Properties

☐ Displays the soil properties for the selected soil or soil line.

The View Soil Properties command allows you to graphically select a soil line or region and view the properties associated with the soil or display a list of all soil properties. The soil properties can be printed or copied to the Windows clipboard for importing into other applications.

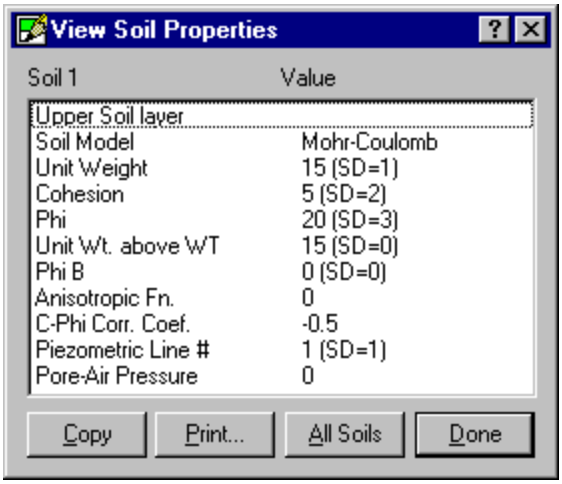
NOTE: Use the Sketch Text command to place the soil properties on the drawing as a label. This allows you to print the soil properties on the drawing for reference purposes. If you change the soil properties using KeyIn Soil Properties, the label will be automatically updated with the new soil properties.

➢ **To view the soil properties:**

1. Choose the Soil Properties command from either the View menu or from the Mode Toolbar.

   The cursor changes from an arrow to a cross-hair, the status bar indicates that “View Soil Properties” is the current operating mode, and an empty dialog box is displayed.

2. Move the cursor inside the desired soil layer or on top of the soil line and click the left mouse button. The soil is selected with a diagonal hatch pattern, and the soil line and points are highlighted. The soil properties are displayed in the dialog box as follows:



   The dialog box lists the soil number, description, model, the properties specific to the soil model, any piezometric line or r_u value defined for the soil, and the pore-air pressure. For probabilistic analyses, the standard deviation is listed after the mean value of each soil property.

3. To see all the soil properties, re-size the dialog box by dragging the bottom edge of the window down until all information is displayed.

4. Repeat Step 2 for every soil that you wish to view.

5. To view a list of all soil properties in the dialog box, select the All Soils button.

   The currently-selected soil is unselected, and all soil properties are displayed.

6. To copy the soil properties to the Windows Clipboard, select Copy. The soil properties are copied to the Clipboard in the following text format:

```
Soil 1
Upper Soil layer
Soil Model                  Mohr-Coulomb
Unit Weight                 15 (SD=1)
Cohesion                    5 (SD=2)
Phi                         20 (SD=3)
Unit Wt. above WT           15 (SD=0)
Phi B                       0 (SD=0)
Anisotropic Fn.             0
C-Phi Corr. Coef.           -0.5
Piezometric Line #          1 (SD=1)
Pore-Air Pressure           0
```

7. To print the soil properties on the current printer, select Print. The soil properties will be printed in the same format as was used for copying to the Clipboard.

8. Select Done, press ESC, or click the right mouse button to finish viewing soil properties.

Comments

To change the soil properties, choose KeyIn Soil Properties. To change the pore-air pressure, choose KeyIn Pore Pressure: Air Pressure. To change the piezometric line or r_u value, choose Draw Pore-Water Pressure.

See the View Point Information command for information on displaying point information.

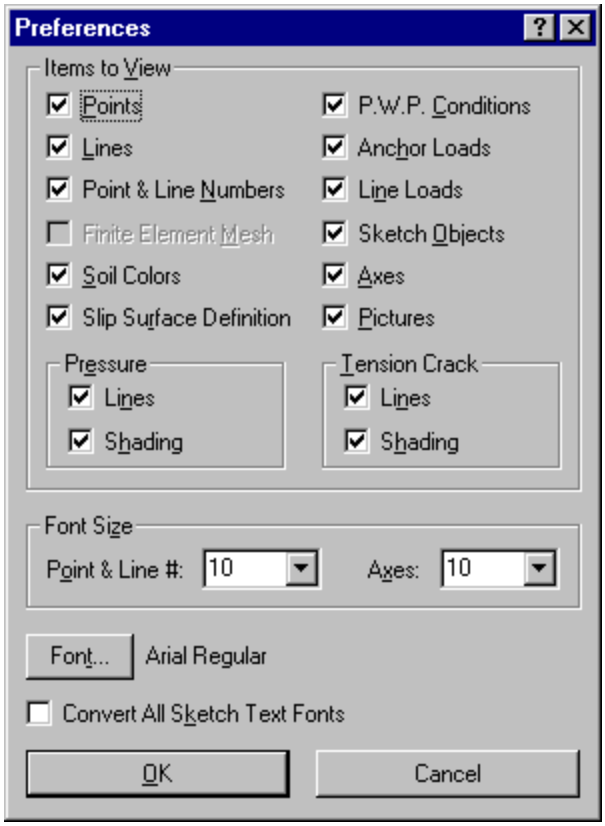
View Preferences

☐ Identifies which items will be displayed on the drawing.

The View Preferences command allows you to display different types of objects on the drawing at the same time. All object types are displayed by default; however, you can turn off object types that you do not wish to view.

This command also can be used to change the default font used for the problem, as well as the font size used for point and line numbers and for the axes. The default font is used for all text in the problem except text items created with Sketch Text.

When you choose View Preferences, the following dialog box is displayed:



NOTE: The View Preferences toolbar also provides access to the View Preferences dialog box. The toolbar is usually more convenient to use than the View Preferences menu command, since it also provides a toolbar button for each item type to view. This allows you to change the item types displayed on the drawing while you are using another command, such as Modify Objects.

➢ **To select the items to view:**

- In the Items To View group box, check the items that you want displayed on the drawing. Any items that are cleared will remain in the problem definition but will not be displayed.

Points Displays points as small squares.

Lines Displays soil geometry lines.

Point & Line Numbers Displays point and line numbers only if points or lines are also displayed.

Finite Element Mesh Displays the finite element mesh imported from a SEEP/W or SIGMA/W data file.

Soil Colors Displays soil layers as different colors, depending on the soil color assigned to the soil number.

Slip Surface Definition Displays the slip surface grid and radius, the complete fully specified slip surfaces, or the left and right slip surface block grids. The slip surface axis point and the slip surface limits are also displayed.

P.W.P. Conditions Displays pore-water pressure conditions. Piezometric lines and contours are displayed as blue dashed lines. Pore-water conditions at points are displayed as triangles. Other conditions, such as r_u values, are not graphically displayed.

Anchor Loads Displays anchor loads as a line segment with an arrow pointing in the direction of the anchor load. The bonded portion of the anchor is shown as a thick line.

Line Loads Displays line loads as small arrows pointing in the direction of the load.

Sketch Objects Displays text, lines, circles, and arcs created by the Sketch commands.

Axes Displays the axes.

Pictures Displays imported bitmap or metafile pictures.

Pressure Displays surface pressure lines and/or shading. If Shading is selected, the area between the pressure line and the top soil surface is shaded with a cross-hatch pattern.

Tension Crack Displays the tension crack line and/or shading. If Shading is selected, the area between the tension crack line and the top soil surface is shaded with vertical-line pattern.

Font Sizes

Point numbers, soil line numbers, and axis numbers are displayed at the point sizes listed in the Font Size group box.

➢ **To change a font size:**

- Click the down arrow to the right of the Point & Line # or Axes edit boxes and select a point size from the list, or type the desired point size in the edit box..

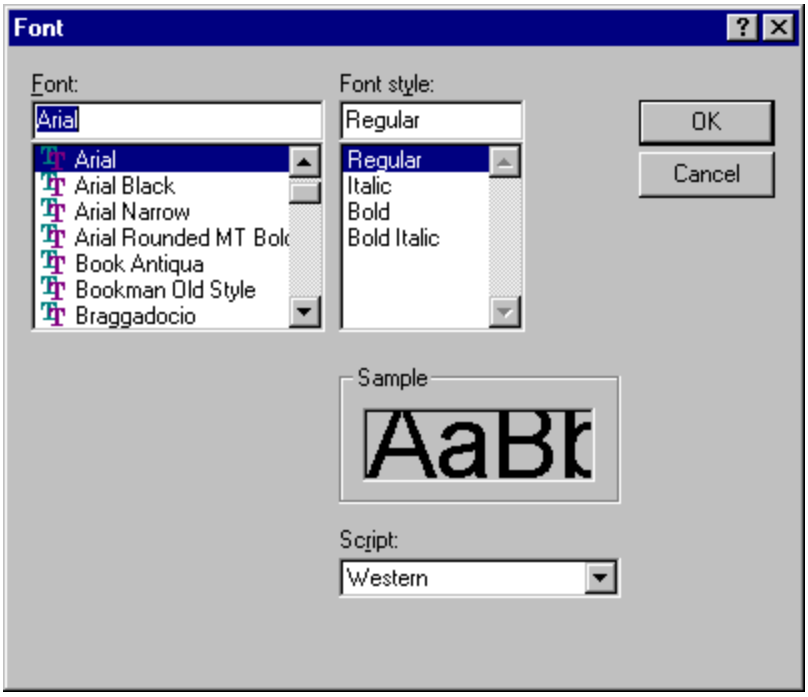
Points are the units commonly used for font size (72 points is equal to 1inch). The point size that you enter represents the height of the point or axis numbers at a zoom factor of 1.0.

Default Font

SLOPE/W uses the default font to display point numbers, soil line numbers, axis numbers, axis labels, and function graph numbers and labels.

➢ **To change the default font:**

1. Click on the Font button. The following dialog box is displayed:



All the fonts that are currently installed in Windows are displayed in the Font list box. To install or delete fonts, you must use the Windows Control Panel. See the Windows User's Guide for more information on Control Panel.

2. Select the desired font in the Font list box and style in the Font Style list box.

3. Select OK to return to the View Preferences dialog box. The name of the selected font is displayed beside the Font button.

NOTE: SLOPE/W does *not* use the default font to display sketch text on the drawing. Therefore, when you select a new default font, all text defined with the Sketch Text command remains unchanged. This is undesirable if you wish to use one font for all text that appears on the drawing.

➢ **To change the font for *all* sketch text to the default font:**

1. Select the Convert All Sketch Text Fonts check box.

2. When you select the OK button in the View Preferences dialog box, the program asks if you wish to change all sketch text fonts to the default font.

3. Select Yes to change all sketch text fonts to the default font; select No to exit the View Preferences dialog box without changing the sketch text fonts; or select Cancel to return to the View Preferences dialog box.

Comments

Only the items displayed are shown on paper when you print the drawing. This allows you to print any combination of items on your drawing.

When you define an item, SLOPE/W will check the item in View Preferences if you have not already checked it. For example, if you choose Draw Points, SLOPE/W will check the Points option in View Preferences. This enables you to see the points that you define

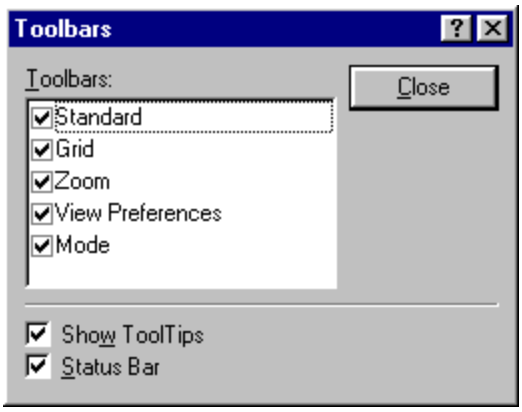
View Toolbars

☐ Displays or hides the DEFINE toolbars and the status bar.

Use the View Toolbars command to toggle the display of any toolbar, the status bar, or the toolbar tool tips.

➢ **To change the toolbar and status bar display:**

1. Select the Toolbars command from the View menu or right-click on a toolbar and select Toolbars from the pop-up context menu. The following dialog box appears:



2. In the Toolbars list box, check the toolbars you wish to display, or uncheck the toolbars you wish to hide by clicking on the check boxes with the left mouse button.

   Each time you check an item, it appears in the DEFINE window; each time you uncheck an item, it is removed from the DEFINE window.

3. To show or remove the tool tips that are displayed when the mouse is over a toolbar button, check or uncheck the Show ToolTips check box.

4. To show or remove the status bar from the bottom of the DEFINE window, check or uncheck the Status Bar check box. The information displayed in the status bar is described below.

5. When finished, click on the Close button.

NOTE: You can quickly add or remove a toolbar or status bar by clicking the right mouse button on top of any toolbar or status bar. When the pop-up menu appears, select a toolbar or the status bar from the menu to toggle its display.

Status Bar

The status bar contains three panes and is displayed as follows:

Status Information Current status of the program. If the mouse cursor is above a menu item or toolbar button, the purpose of the menu item or toolbar button is displayed. If the program is in a "mode", then the current mode and suggested user action is displayed. The status bar above is shown in the default mode.

Mouse Coordinates Mouse cursor coordinates in engineering units.

View Redraw

☐ Redraws the problem.

View Redraw clears the DEFINE window and re-displays the drawing in the window. This is sometimes needed when drawing objects or when you are scrolling, since objects may not be completely drawn in the window.

The KeyIn Menu

The KeyIn menu commands are:

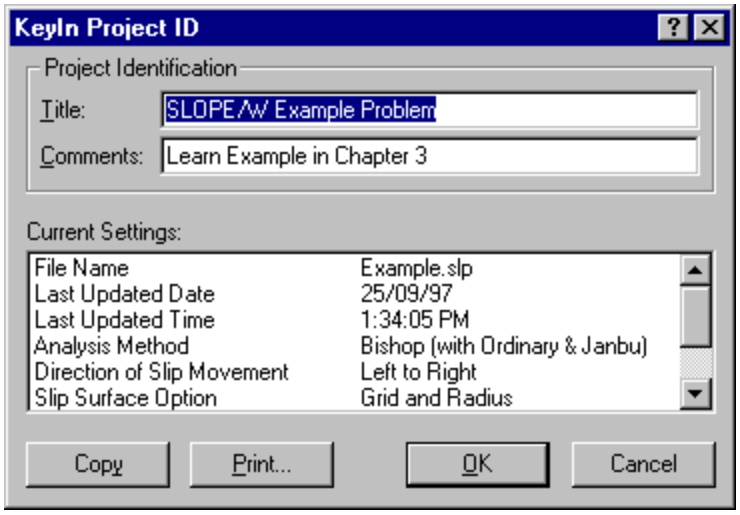
- ***Project ID*** Identifies the problem and displays information about the selected options. For more information about this command, see KeyIn Project ID in this chapter.
- ***Analysis Method*** Selects the method of analysis and the interslice force function. For more information about this command, see KeyIn Analysis Method in this chapter.
- ***Analysis Control*** Specifies the probability, convergence, slip surface, pore-water pressure, and tension crack options. For more information about this command, see KeyIn Analysis Control in this chapter.
- ***Soil Properties*** Defines the soil properties. For more information about this command, see KeyIn Soil Properties in this chapter.
- ***Strength Functions: Shear/Normal*** Defines the relationship between shear stress and normal stress for the soil properties. For more information about this command, see KeyIn Strength Functions: Shear/Normal in this chapter.
- ***Strength Functions: Anisotropic*** Defines the relationship between soil strength and the slice inclination angle. For more information about this command, see KeyIn Strength Functions: Anisotropic in this chapter.
- ***Tension Crack*** Defines the tension crack line or angle. For more information about this command, see KeyIn Tension Crack in this chapter.
- ***Points*** Defines points used in specifying the geometric data. For more information about this command, see KeyIn Points in this chapter.
- ***Lines*** Defines the boundaries for each soil. For more information about this command, see KeyIn Lines in this chapter.
- ***Slip Surface: Grid & Radius*** Defines the rotation centers and radii for circular and composite slip surfaces. For more information about this command, see KeyIn Slip Surface: Grid & Radius in this chapter.
- ***Slip Surface: Axis*** Selects the point about which to compute moment equilibrium. For more information about this command, see KeyIn Slip Surface: Axis in this chapter.
- ***Slip Surface: Specified*** Defines the slip surfaces as piece-wise linear line segments. For more information about this command, see KeyIn Slip Surface: Specified in this chapter.
- ***Slip Surface: Left Block*** Defines the left block of intersection points for a generated piece-wise linear slip surface. For more information about this command, see KeyIn Slip Surface: Left Block in this chapter.
- ***Slip Surface: Right Block*** Defines the right block of intersection points for a generated piece-wise linear slip surface. For more information about this command, see KeyIn Slip Surface: Right Block in this chapter.
- ***Slip Surface: Limits*** Defines the limits within which the slip surface must intersect the top soil layer. For more information about this command, see KeyIn Slip Surface: Limits in this chapter.
- ***Pore Pressure: Water Pressure*** Defines the pore-water pressure conditions. For more information about this command, see KeyIn Pore Pressure: Water Pressure in this chapter.
- ***Pore Pressure: Air Pressure*** Specifies pore-air pressure for each soil layer. For more information about this command, see KeyIn Pore Pressure: Air Pressure in this chapter.

- ***Load: Line Loads*** Sets the position, magnitude, and direction of concentrated loads. For more information about this command, see KeyIn Load: Line Loads in this chapter.
- ***Load: Anchor Loads*** Defines anchors acting as concentrated loads within the soil. For more information about this command, see KeyIn Load: Anchor Loads in this chapter.
- ***Load: Seismic Load*** Sets horizontal and vertical coefficients representing a seismic force. For more information about this command, see KeyIn Load: Seismic Load in this chapter.
- ***Pressure Lines*** Defines pressure lines applied at the top soil surface. For more information about this command, see KeyIn Pressure Lines in this chapter.

KeyIn Project ID

☐ Identifies the problem and displays information about the selected options.

When you choose KeyIn Project ID, the following dialog box appears:



The Project Information group box allows you to enter the following information:

Title The title of the problem.

Comments Other information about the problem.

➢ **To identify the problem:**

- Type any text in the Title and Comments edit boxes. This information is saved as an identifying header in all output files created by SLOPE/W SOLVE. It is also placed on the drawing when you choose Sketch Text and add a Project ID label.

The Current Settings group box displays the current values of the following problem parameters:

File Name The name of the problem data file.

Last Updated Date The last date at which the problem was saved to the data file.

Last Updated Time The last time at which the problem was saved to the data file.

Analysis Method The analysis method selected using KeyIn Analysis Method.

Direction of Slip Movement The direction in which the slip surface may move, as specified using KeyIn Analysis Control.

Slip Surface Option The way in which the slip surface is defined, as specified using KeyIn Analysis Control.

P.W.P. Option The way in which the pore-water pressure is defined, as specified using KeyIn Analysis Control.

Tension Crack Option The way in which a tension crack is defined, as specified using KeyIn Analysis Control.

Seismic Coefficient The directions in which a seismic coefficient is defined (horizontal and/or vertical) using KeyIn Load: Seismic Load.

➢ **To copy the Project ID information to the Windows Clipboard:**

- Select Copy. The Project Identification information and Selected Options are copied to the Clipboard in the following text format:

```
SLOPE/W Example Problem
Learn Example in Chapter 3
File Name                   Example.slp
Last Updated Date           25/09/97
Last Updated Time           1:34:05 PM
Analysis Method             Bishop (with Ordinary & Janbu)
Direction of Slip Movement  Left to Right
Slip Surface Option         Grid and Radius
P.W.P. Option               Piezometric Lines / Ru
Tension Crack Option        (none)
Seismic Coefficient         (none)
```

➢ **To print the Project ID information:**

- Select Print. The Project ID information will be printed in the same format as for copying to the Clipboard.

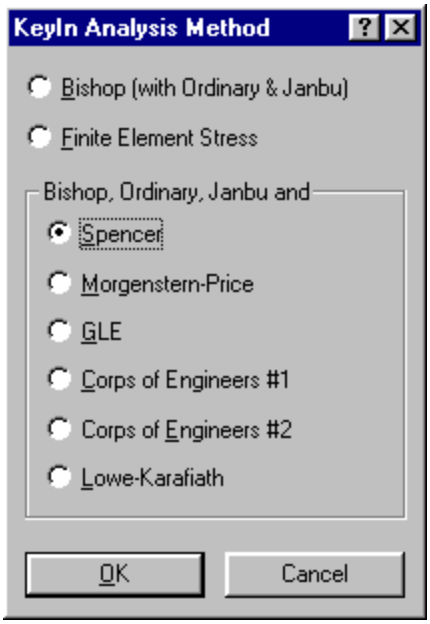
➢ **To place a Project ID label on the drawing:**

- Choose the Sketch Text command and select the Project ID tab in the dialog box. The label you add on the drawing is automatically updated whenever any of the project information changes.

KeyIn Analysis Method

☐ Selects the method of analysis and the interslice force function.

When you choose KeyIn Analysis Method, the following dialog box appears:



➢ **To set the analysis method:**

1. Select the analysis method.
2. Select OK.

SLOPE/W always calculates the factors of safety for the Bishop, Ordinary, and Janbu methods as a group. None of these three methods can be selected individually.

Analysis Methods:

Bishop (with Ordinary & Janbu) Method

- The Ordinary method sets both the normal and shear interslice forces to zero.
- The Bishop and Janbu methods consider normal forces but not shear forces between the slices.
- The Bishop method satisfies only moment equilibrium, and the Janbu method satisfies only force equilibrium.
- The Janbu factor of safety does not include Janbu's empirical correction factor, f_o. The correction factor must be manually applied. See the Theory section for further information on the correction factor.

Finite Element Stress Method

- The Finite Element Stress method uses SIGMA/W computed stresses to calculate stability factor.
- The SIGMA/W stresses s_x, s_y, and t_{xy} are used to compute the normal stress and mobilized shear stress at the base of each slice. The base normal is in turn used to calculate the available shear strength.
- The summation of the available shear strength along the entire slip surface is divided by the summation of the corresponding mobilized shear stress to establish the stability factor.

Spencer Method

- Spencer's method satisfies both force and moment equilibrium and is restricted to a constant interslice force function.

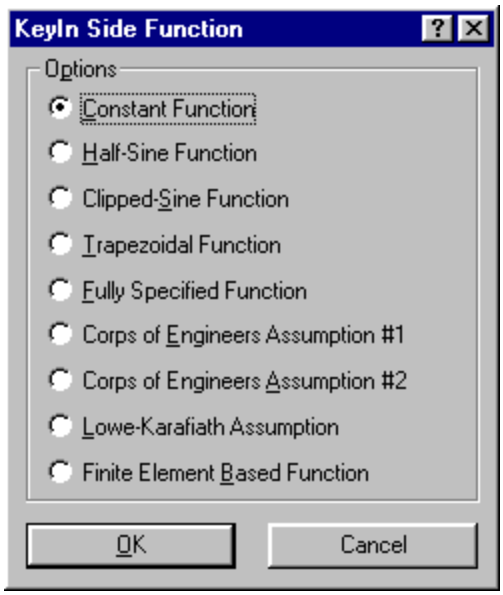
- SLOPE/W uses the "Rapid Solver" technique, as described in the Theory section, to compute the l value that results in the same factor of safety for both moment and force equilibrium.

Morgenstern-Price Method

- The Morgenstern-Price method satisfies both force and moment equilibrium and uses a selected interslice force function.

- SLOPE/W uses the "Rapid Solver" technique, as described in the Theory section, to compute the lambda (l) value that results in the same factor of safety for both moment and force equilibrium.

When you select the Morgenstern Price method, the following dialog box appears:



➢ **To select the side function for the Morgenstern Price method:**

1. Select the appropriate interslice force function.

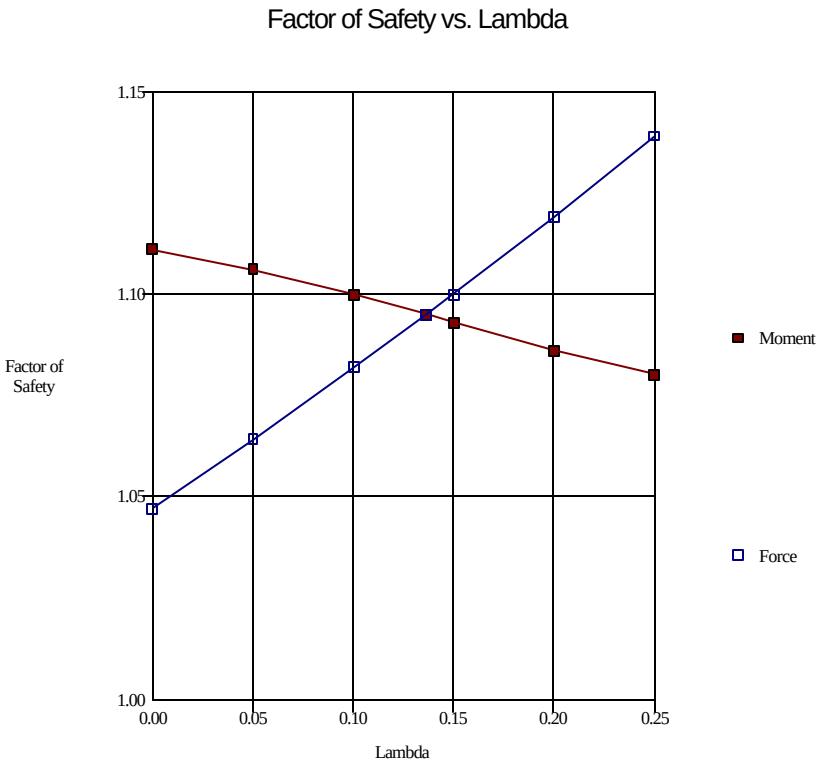
2. Select OK.

These functions are discussed further on in this section.

NOTE: Selecting the Morgenstern - Price method with a constant interslice force function is the same as selecting Spencer's method.

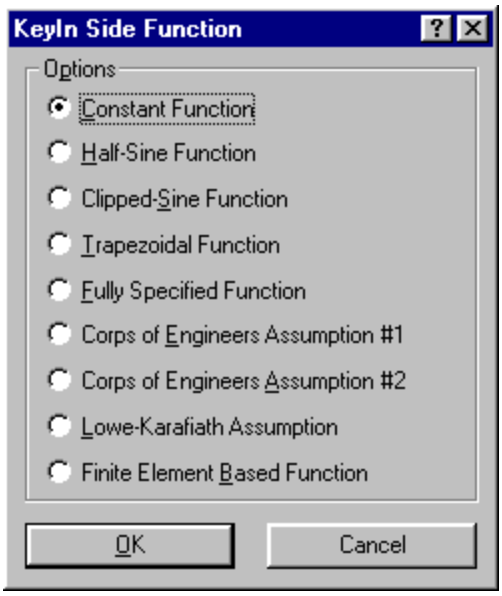
General Limit Equilibrium (GLE) Method

- The GLE method is much like the Morgenstern-Price method, except that it can be used to compute the moment and force factors of safety for a range of user-defined lambda (l) values. A plot of factor of safety versus lambda can be created using CONTOUR.



- The GLE method satisfies both force and moment equilibrium and uses a selected interslice force function.
- SLOPE/W uses piece-wise linear line segments between the data points to find the intersection point of the two lines.
- When the range of lambda (l) values is such that the two lines do not cross, SLOPE/W is unable to compute a solution.

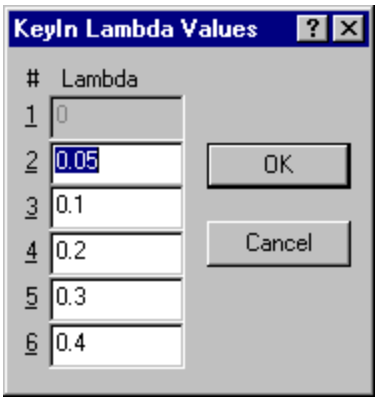
When you select the GLE method, the following dialog box appears:



➢ **To select the side function for the GLE method:**

1. Select the appropriate interslice force function.
2. Select OK.

   The following dialog box appears:



The default values for lambda (0, 0.05, 0.1, 0.2, 0.3, and 0.4) appear in the six edit boxes. If you wish to change these values, specify six positive or negative values for lambda. In a left-to-right problem, the cross-over lambda value is usually positive, while in a right-to-left problem, the cross-over lambda value is usually negative. While this generally is the case, high lateral loads can cause the reverse to be true. The best way to determine the correct range of lambda values is to first perform an analysis using the Morgenstern-Price method, which uses the "Rapid Solver" technique to get the correct lambda value. You can then use this value to set an appropriate range of lambda values for the GLE method.

Corps of Engineers #1 Method

- The Corps of Engineers #1 method satisfies only force equilibrium.

- The special interslice force function is computed by SLOPE/W. The direction of the interslice force for each slice is set equal to the average surface slope, as described in the Theory section.

- Lambda is always equal to 1.0.

Corps of Engineers #2 Method

- The Corps of Engineers #2 method satisfies only force equilibrium.

- The special interslice force function is computed by SLOPE/W. The direction of the interslice force for each slice is set equal to the ground surface slope at the top of each slice, as described in the Theory chapter.

- Lambda is always equal to 1.0.

Lowe-Karafiath Method

- The Lowe-Karafiath method satisfies only force equilibrium.

- The special interslice force function is computed by SLOPE/W. The direction of the interslice force is set equal to the average of the ground surface slope at the top of the slice and the slip surface slope at the bottom of the slice. This function is described in the Theory section.

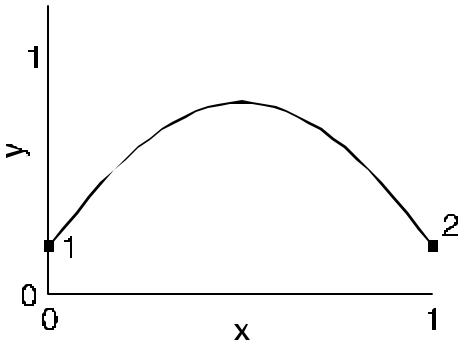
- Lambda is always equal to 1.0.

Interslice Force Function Types:

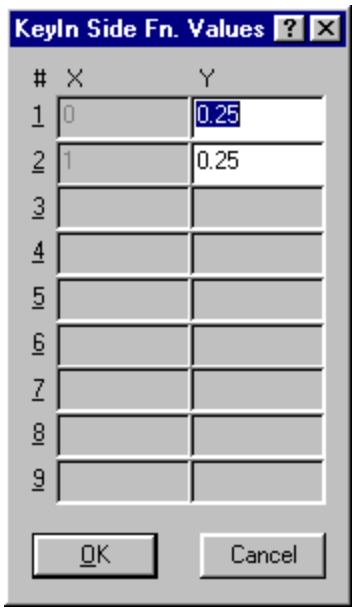
The Morgenstern-Price and GLE methods let you define an interslice force function. For the following functions, you must specify the *x*- and *y*-coordinates of the functions.

Clipped-Sine Function

- When using the clipped-sine function, you must specify the *y*-coordinates when *x* is equal to 0.0 and *x* is equal to 1.0.



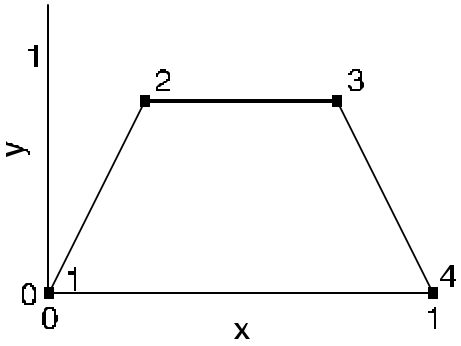
When you select the clipped-sine function, the following dialog box appears:



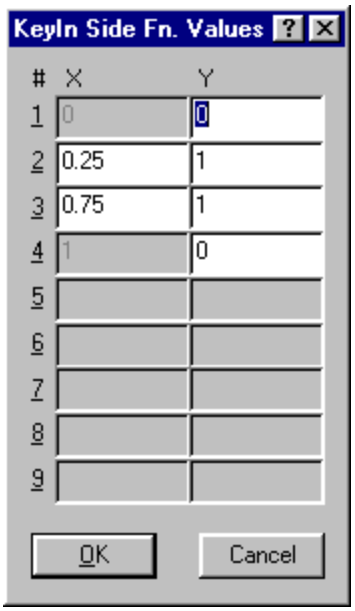
The default *y*-coordinate values (0.25) appear in the two edit boxes. If you wish to change these values, specify two *y*-coordinate values between 0.0 and 1.0.

Trapezoidal Function

- When using the trapezoidal function, you must specify four *x*- and *y*-coordinates.



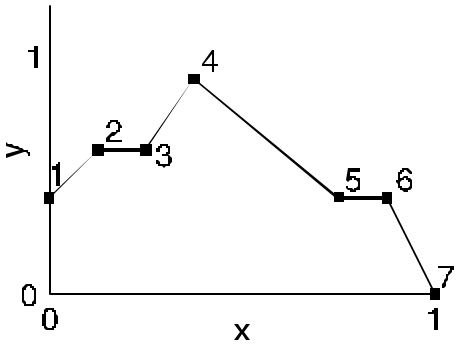
When you select the trapezoidal function, the following dialog box appears:



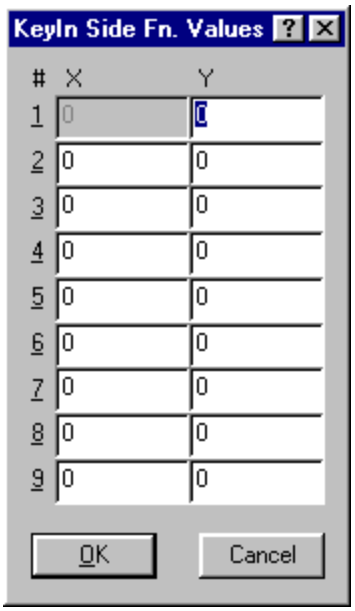
The default coordinate values appear as displayed in the above dialog box. If you wish to change these values, specify four points. The *x*-coordinate of the first point must be 0.0, and the *x*-coordinate of the fourth point must be 1.0. All other values must be between 0.0 and 1.0.

Fully Specified Function

- You can specify any piece-wise linear side force function with up to nine points.



When you select the fully specified function, the following dialog box appears:

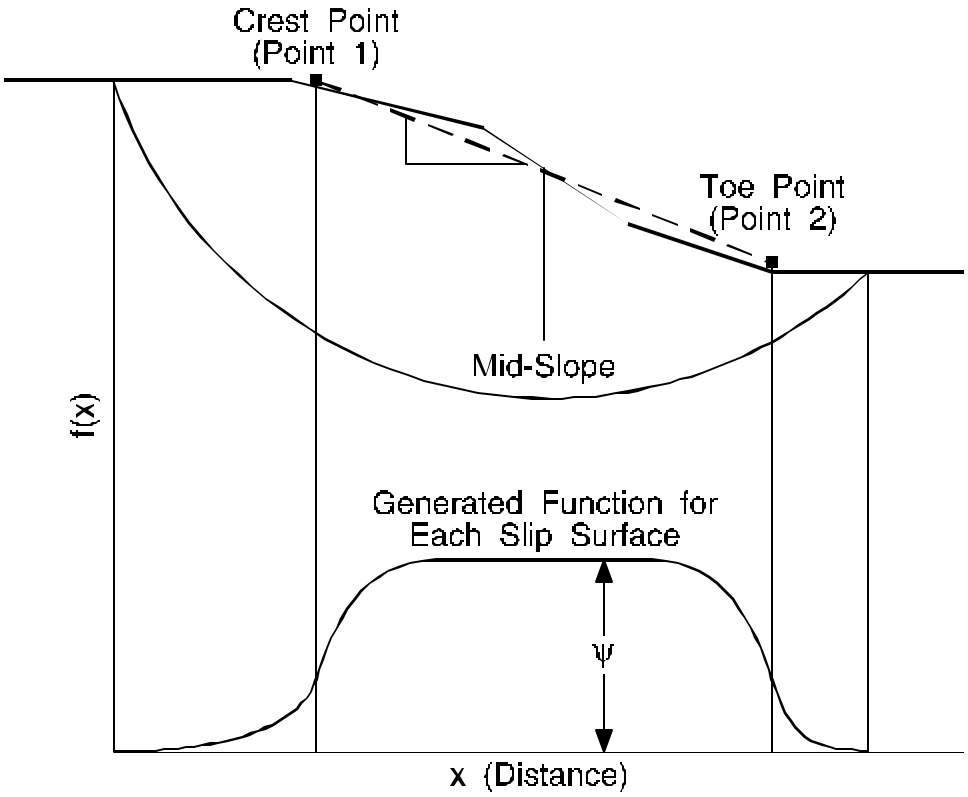


The default coordinate values are all equal to 0 (no points are defined). To define points, specify the *x*-coordinate of the first point as 0.0 and the *x*-coordinate of the last point as 1.0. All other *x*- and *y*-coordinates must be between 0.0 and 1.0. Any unused points must have all coordinates specified as 0. For example, if you define five points, the *x*-coordinate for Point 1 should be 0.0, the *x*-coordinate for Point 5 should be 1.0, and the *x*- and *y*-coordinates for Points 6, 7, 8, and 9 should all be 0.

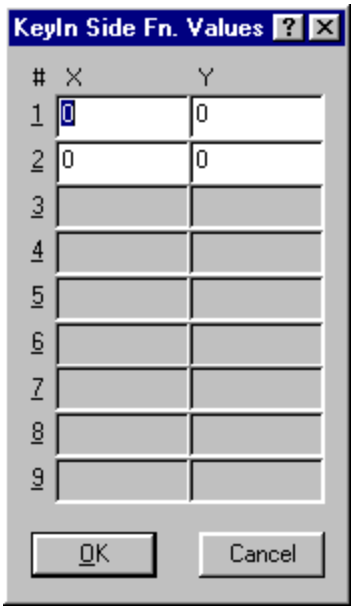
Finite Element Based Function

The Theory chapter discusses the background to the finite element based function. To use this function, it is necessary to specify the *x*- and *y*-coordinates of the slope crest and toe, as shown in Figure 4.7.

Figure 4.7 Relationship of Finite Element Function to the Slope Geometry



When you select the finite element based function, the following dialog box appears:



➢ **To specify the finite element based side function:**

1. Specify the crest and toe *x*- and *y*-coordinates as Points 1 and 2.
2. Select OK.

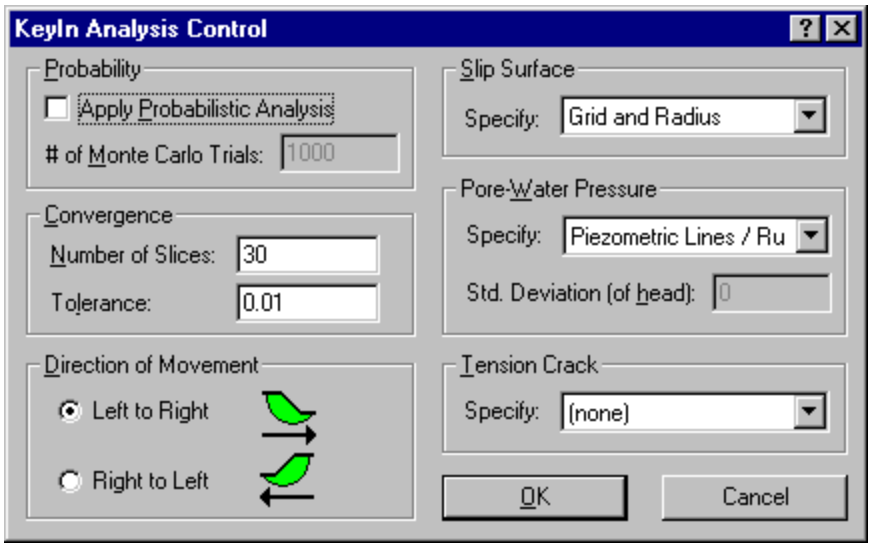
NOTE: The *x*- and *y*-coordinates for Points 1 and 2 are the actual geometric coordinates of the crest and toe.

For irregular slopes, you must exercise some judgment when selecting the coordinates. Remember that the crest and toe coordinates are used only for the purpose of defining the inflection points of the finite element based function.

KeyIn Analysis Control

☐ Specifies the probability, convergence, slip surface, pore-water pressure, and tension crack options.

When you choose KeyIn Analysis Control, the following dialog box appears:



The following analysis control options are available for selection:

Probability Options:

Apply Probabilistic Analysis Selecting this check box indicates that you will be performing a probabilistic slope stability analysis. When this option is chosen, you can enter the variability of certain input parameters, such as soil properties, using a standard deviation (S.D.). The default value of all standard deviations is zero.

# of Monte Carlo Trials You must specify how many Monte Carlo trials SLOPE/W should perform. The default number of Monte Carlo trials is 1000. See Probabilistic Analysis in the Modelling Guidelines chapter and Probabilistic Slope Stability Analysis in the Theory chapter for further discussion on how SLOPE/W performs probabilistic analyses.

Convergence Options:

Number of Slices This parameter defines the minimum number of slices in the sliding mass. The default value is 30, and is adequate for most problems.

Tolerance This parameter is the desired difference in factor of safety between any two iterations. When the difference in factor of safety between two iterations is less than the tolerance, the solution has converged and the iteration process stops. The default value is 0.01.

Direction of Movement Options:

Left to Right With this option, each slip surface is assumed to move from the left side of the problem to the right side for purposes of calculating the factor of safety. SLOPE/W considers the movement to be left-to-right when the crest entrance point of the slip surface is higher and to the left of the toe exit point. If this entrance point is lower in elevation than the exit point, SLOPE/W bypasses the slip surface.

Right to Left With this option, each slip surface is assumed to move from the right side of the problem to the left side for purposes of calculating the factor of safety. SLOPE/W considers the movement to be right-to-left when the crest entrance point of the slip surface is higher and to the right of the toe exit point. If this entrance point is lower in elevation than the exit point, SLOPE/W bypasses the slip surface.

Slip Surface Options:

Grid and Radius With this option, the slip surface is defined by a rotation center, a radius, the arc of a circle, and straight line segments if the circle encounters an impenetrable layer, a tension crack, a projection angle, or water. The grid of rotation centers and the radius lines are defined by choosing KeyIn Slip Surface Grid & Radius, Draw Slip Surface Grid, or Draw Slip Surface Radius.

Fully Specified With this option, the slip surface must be fully specified by a series of straight line segments. The line segments are defined by choosing KeyIn Slip Surface: Specified or Draw Slip Surface Specified. The slip surface axis point is defined by choosing KeyIn Slip Surface Axis or Draw Slip Surface Axis.

Block Specified With this option, the slip surface consists of several line segments defined by two grids of intersection points. Slip surfaces are created by connecting each point in the left block with each point in the right block, and then projecting each point to the surface using the specified range of projection angles. The left and right blocks are defined by choosing KeyIn Slip Surface Left Block, KeyIn Slip Surface Right Block, Draw Slip Surface Left Block, or Draw Slip Surface Right Block. The slip surface axis point is defined by choosing KeyIn Slip Surface Axis or Draw Slip Surface Axis.

Pore-Water Pressure Options:

(none) Pore-water pressure is not taken into account when computing the factor of safety.

R_u ***Coefficients*** An r_u value must be specified for each soil type. The r_u coefficient is defined as:

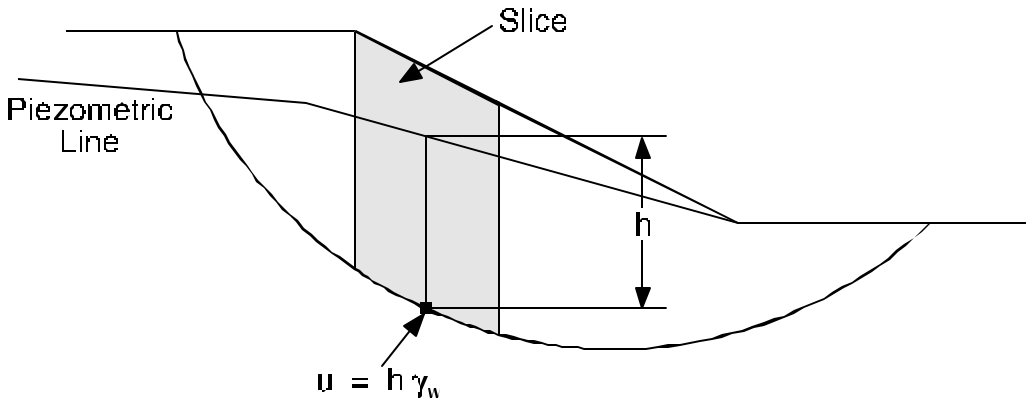
$$r_u = \frac{u}{\sum \gamma_i h_i} \tag{4.1}$$

where:

γ_i = the total unit weight of each soil strata in the slice
h_i = the average thickness of each soil strata in the slice
i = the soil strata number
u = pore-water pressure

Piezometric Lines / R_u When a piezometric line is specified, SLOPE/W computes the pore-water pressure at the base of the slice as the vertical distance from the centroid of the slice base to the piezometric line multiplied by the unit weight of water. Figure 4.8 shows how the pore-water pressure is specified.

Figure 4.8 Specification of Pore-Water Pressure Using a Piezometric Line



Piezometric lines can be used in conjunction with r_u coefficients within the same problem. The piezometric and r_u effects can be combined or used independently for different soil types. When the piezometric line is below the slice base, a negative pore-water pressure will be computed and used if the unsaturated soil strength parameter f^b is nonzero. Negative pore-water pressures are set to zero when f^b is zero.

NOTE: A piezometric line is not required for soil with no strength (that is, c=0.0 and f=0.0), such as water impounded against the slope.

A piezometric line is not required for a soil layer with a tension crack, since the slip surface is vertical throughout such a layer.

A piezometric line is not required for materials designated as bedrock.

Pore-Water Pressure Contours The pore-water pressures can be defined by a series of constant pressure contours. The pore-water pressure at the base of a slice is computed by linear extrapolation along a vertical line on the bases of the nearest two contours. Each contour line must start at the left extremity of the problem and extend across the problem to the right extremity. The starting and ending x-coordinates of each contour line must match the left and right boundaries of the problem and the pressure contour values may be positive or negative.

Grid of Heads Pore-water pressure head may be specified at discrete points. The head must be specified as pressure head, such as meters of water or feet of water. SLOPE/W multiplies the head by the unit weight of water to convert the head into pressure and spline interpolation is used to establish the pore-water pressure at the base of each slice.

Grid of Pressures Pore-water pressure may be specified at discrete points. Spline interpolation is used to establish the pore-water pressure at the base of each slice.

Grid of R_u Coefficients Pore-water pressure may be specified by defining the r_u coefficient at discrete points. Spline interpolation is used to establish the pore-water pressure at the base of each slice.

SEEP/W Heads The pore-water pressure heads computed by a SEEP/W finite element seepage analysis may be used in a SLOPE/W analysis. When you select the SEEP/W Heads option, the SLOPE/W SOLVE function will prompt you for the SEEP/W file containing the desired head values.

NOTE: The SEEP/W Heads option can only be selected if you have imported a finite element mesh using File Import: Data File.

SIGMA/W Pressures The pore-water pressures computed by a SIGMA/W finite element stress/deformation analysis may be used in a SLOPE/W analysis. When you select the SIGMA/W Pressures option, the SLOPE/W SOLVE function will prompt you for the SIGMA/W file containing the desired pore-water pressure values.

NOTE: The SIGMA/W Pressures option can only be selected if you have imported a finite element mesh using File Import: Data File.

Std. Deviation (of head) In a probabilistic analysis, you may specify the standard deviation of the pore water pressure in terms of pore water pressure head. For example, if you are using feet as the unit for length, 2.0 means that the standard deviation of the pore water pressure is 2 feet. In other words, there is a 68% chance that the pore-water pressure head will lie within plus or minus 2 feet of the mean pore water pressure.

Tension Crack Options:

(none) With this option, a tension crack is not specified for the problem.

Tension Crack Angle With this option, a tangent angle is specified on the circular slip surface. When the circular surface reaches the tangent angle point, a tension crack will occur, resulting in the slip surface being projected vertically to the top soil surface. This option can only be selected for Grid & Radius slip surfaces.

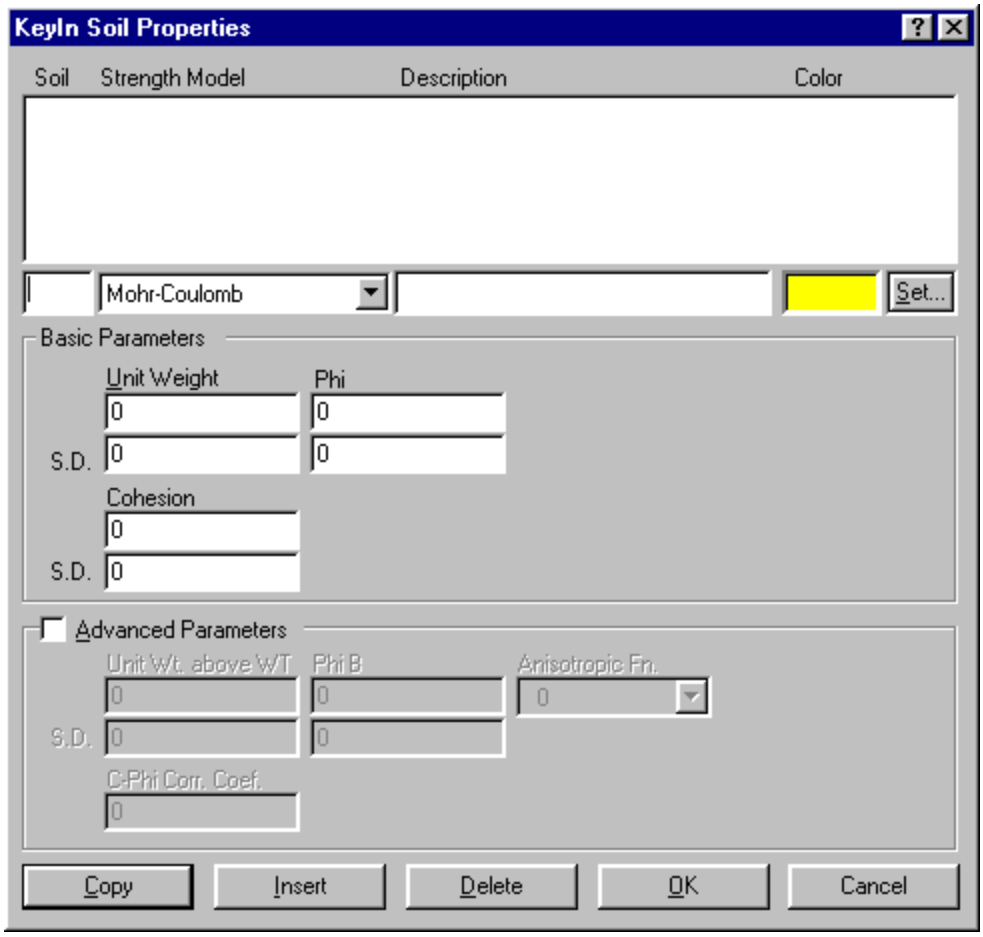
Tension Crack Line With this option, a tension crack line is specified for the problem. When the slip surface intersects the tension crack line, the slip surface is projected vertically to the top soil surface. This option can be used in conjunction with any of the slip surface options.

KeyIn Soil Properties

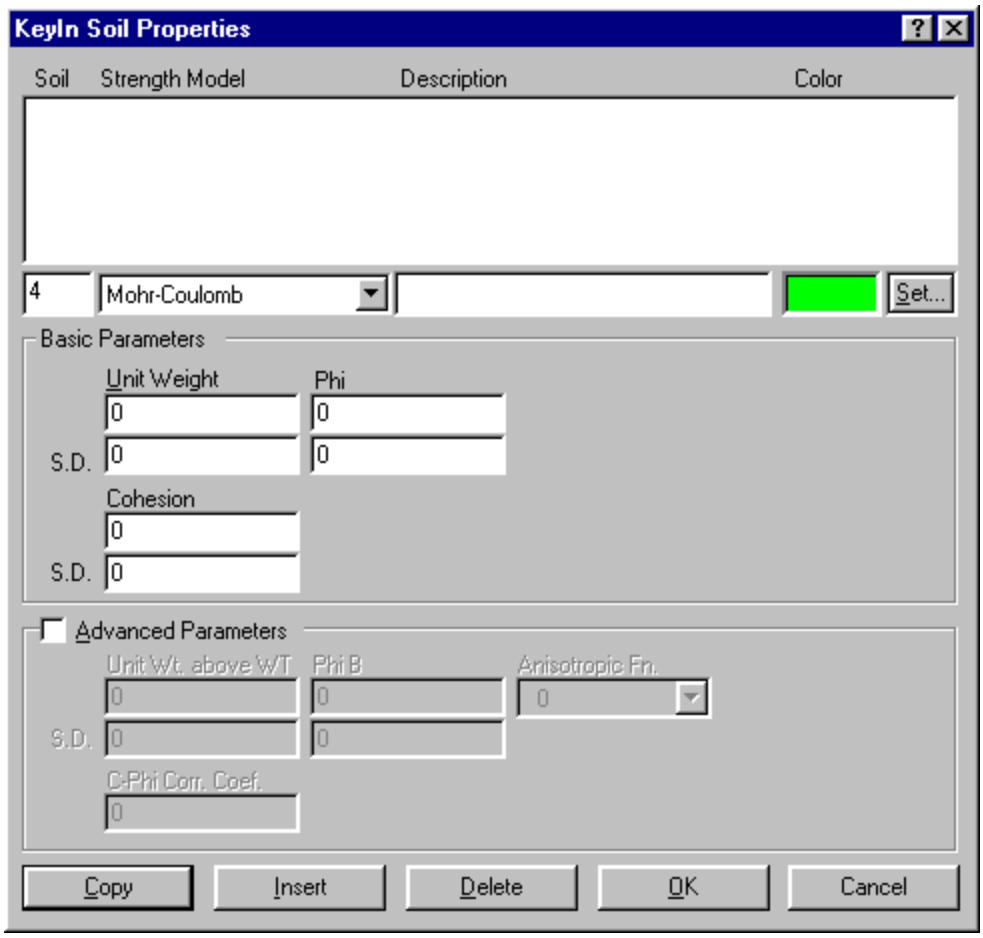
☐ Defines the soil properties.

The KeyIn Soil Properties command allows you to specify soil properties for each soil line. A soil must be defined by choosing KeyIn Soil Properties before you can create the corresponding soil line using Draw Lines or KeyIn Lines.

When you choose KeyIn Soil Properties, the following dialog box appears if you are not defining a probabilistic analysis:



If you have selected a probabilistic analysis using KeyIn Analysis Control, the following dialog box appears instead:



The probabilistic Soil Properties dialog box allows you to enter a standard deviation for each soil property and a C-Phi correlation coefficient.

Soil The number of each soil is displayed in the list box below this heading.

Strength Model The strength model specifies how the soil strength is defined. The strength models available are Mohr-Coulomb, Undrained (Phi=0), No Strength (e.g., water or surcharge), Bedrock, Bilinear, S=f(depth), S=f(elevation), Anisotropic Strength, Shear/Normal Function, Anisotropic Function, Combined: S=f(depth), Combined: S=f(datum) and S=f(overburden).

Description An optional description of the soil type.

Color The color of each soil is displayed in the list box below this heading. Each soil is assigned a default color. Soils using the No Strength model are defaulted to a light blue color to represent water.

Basic Parameters These are basic properties that you must define in order for the soil model to be valid. The basic parameters for each soil model are described later in this section.

Advanced Parameters These are additional soil properties that may not need to be specified. Select the Advanced Parameters check box if you need to define any of these parameters. The advanced parameters for each soil model are described later in this section.

- **To define soil properties:**

  1. Type the soil number in the # edit box.
  2. Select the strength model used for the soil from the Strength Model drop-down list box.
  3. Type a brief description of the soil in the Description edit box. The soil description can be used to label the soil using the Sketch Text command.
  4. Set the soil color by pressing the Set button and selecting a basic color or defining a custom color.
  5. Enter values for the soil properties in the Basic Properties group box. The types of properties to define will depend on the strength model selected for the soil.
  6. To specify any of the advanced soil properties, select the Advanced Properties check box. The advanced property edit boxes will be enabled, allowing you to enter values for any of these soil properties.
  7. Select Copy.

     The specified values are copied to the Soil Properties list box.

- **To insert a new soil layer in the Soil Properties list box:**

  1. In the # edit box, type the new soil number to insert. There may already be a soil with this number in the Soil Properties list box.
  2. Select the soil strength model and enter the remaining soil information as described previously.
  3. Select Insert.

     The new soil properties are inserted in the Soil Properties list box. For example, if you insert a new Soil 3 into a list box containing Soils 1, 2, 3, and 4, Soil 4 is changed to Soil 5 and Soil 3 is changed to Soil 4 in order to make room to insert the new Soil 3.

- **To delete soils from the Soil Properties list box:**

  1. Select the soil to delete in the Soil Properties list box. To select multiple soils in the list box for deletion, either press the CTRL key and click on each soil to delete or press the SHIFT key and click on the first and last soil to delete.
  2. Select the Delete button.

- **To change the default soil color:**

  1. Select the desired soil in the list box.

     The soil properties and color are copied to the appropriate edit boxes.

2. Click on the Set button, located next to the soil color edit box. The following dialog is displayed:

3. Click on one of the basic or custom colors in the dialog box, or select the Define Custom Colors button to select a different color.

   Click on the help button in the top-right corner of the Color dialog box to get context-sensitive help on any control in the dialog box.

4. Once you have chosen a color, select OK in the Color dialog box.

   The selected color is now displayed in the Soil Properties color edit box.

5. Select Copy.

   The new soil color is copied to the Soil Properties list box.

NOTE: The custom colors that you define in the Colors dialog box are stored in the problem data file when you choose File Save or File Save As and are stored in the Windows registry when you choose File Save Default Settings.

Required Soil Properties

SLOPE/W provides a total of 13 different soil models for you to simulate the shear strength characteristic of a soil. Required soil properties are divided into 2 groups: Basic Parameters and Advanced Parameters. The basic properties are basic parameters required in order for the soil model to be valid. The advanced properties are additional parameters that you may use to modify the soil model. The soil property parameters required for each soil model are presented below:

Mohr-Coulomb Model:

Shear strength is computed based on the Mohr-Coulomb equations, as described in Equations 8.1 and 8.19 of the Theory chapter.

Basic

- ***Unit Weight*** Total unit weight of the soil.
- ***Cohesion*** The cohesion component of the shear strength.
- ***Phi*** Friction angle of the soil.
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Advanced

- ***Unit Wt. Above WT*** Total unit weight of the soil above the water table (i.e., above the zero pressure line).
- ***Unsaturated Phi B*** The rate of shear strength increase with a change in negative pore-water pressure. When f^b is zero, all negative pore-water pressures are set to zero. When f^b is nonzero, the effect of the negative pore-water pressures is included in the analysis.
- ***Anisotropic Function*** A function of the modifier factor versus the base inclination angle of each slice. When this function is defined, the shear strength along the base is multiplied by the modifier factor obtained from the function. You can specify this function using the KeyIn Strength Functions: Anisotropic command.
- ***C-Phi Corr. Coef.*** The correlation coefficient between *c* and Phi when a probabilistic analysis is used. Its value ranges from -1.0 to 1.0. See the Theory chapter for more information on this parameter.
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Undrained (Phi=0) Model:

In this model, shear strength is defined by the cohesion of the soil; therefore, pore water pressure is not considered.

Basic

- ***Unit Weight*** Total unit weight of the soil.
- ***Cohesion*** The cohesion component of the shear strength.
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Advanced

- ***Unit Wt. Above WT*** Total unit weight of the soil above the water table (i.e., above the zero pressure line).
- ***Anisotropic Function*** A function of the modifier factor versus the base inclination angle of each slice. When this function is defined, the shear strength along the base is multiplied by the modifier factor obtained from the function. You can specify this function using the KeyIn Strength Functions: Anisotropic command.
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

No Strength Model:

No shear strength is assumed with this model. When the model is selected, "Water" is entered in the Description edit box; however, you can enter a different description if necessary.

NOTE: Surface water can also be simulated by applying a surface pressure line using Draw Pressure Lines or KeyIn Pressure Lines.

Basic

- ***Unit Weight*** Total unit weight of the fluid.

- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

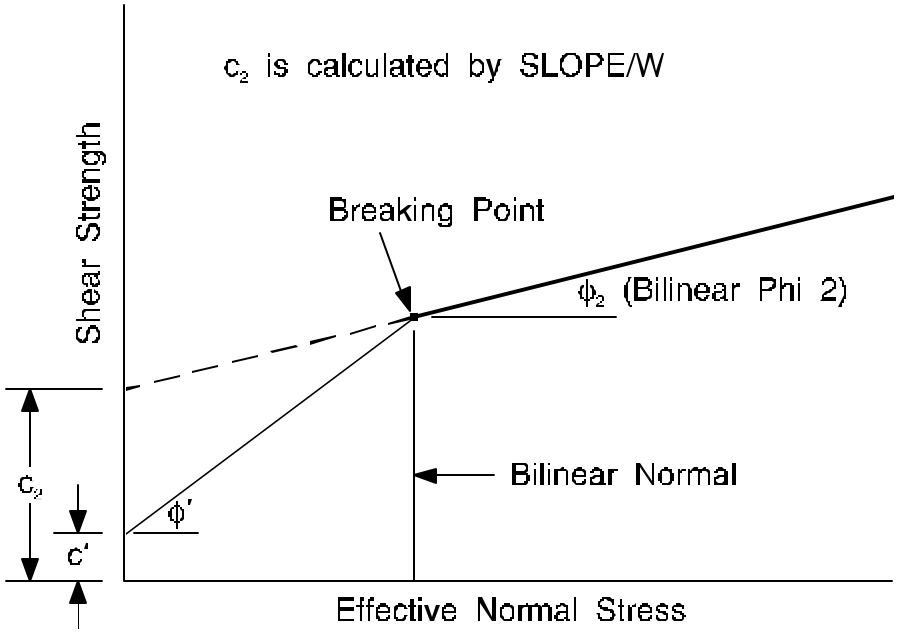
Bedrock Model:

The Bedrock model is used for impenetrable soil. When the model is selected, "Bedrock" is entered in the Description edit box and the Unit Weight is set to -1.0.

Bilinear Model:

The Bilinear model is used to designate a bilinear failure envelope. The definition of the corresponding variables is illustrated in Figure 4.9.

Figure 4.9 Definition of Bilinear Failure Envelope Variables



Basic

- ***Unit Weight*** Total unit weight of the soil.
- ***Cohesion*** The cohesion component of the shear strength.
- ***Phi*** Friction angle of the soil for normal stress smaller than the Normal value.
- ***Phi 2*** Friction angle of the soil for normal stress larger than the Normal value.
- ***Normal*** The normal stress at the breaking point in the failure envelope.
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Advanced

- ***Unit Wt. Above WT*** Total unit weight of the soil above the water table (i.e., above the zero pressure line).
- ***Unsaturated Phi B*** The rate of shear strength increase with a change in negative pore-water pressure. When f^b is zero, all negative pore-water pressures are set to zero. When f^b is nonzero, the effect of the negative pore-water pressures is included in the analysis.

- ***Anisotropic Function*** A function of the modifier factor versus the base inclination angle of each slice. When this function is defined, the shear strength along the base is multiplied by the modifier factor obtained from the function. You can specify this function using the KeyIn Strength Functions: Anisotropic command.

- ***C-Phi Corr. Coef.*** The correlation coefficient between *c* and Phi when a probabilistic analysis is used. Its value ranges from -1.0 to 1.0. See Chapter 8 for more information on this parameter.

- ***Phi-Phi2 Corr. Coef.*** The correlation coefficient between Phi and Phi2 when a probabilistic analysis is used.

- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

S=f(depth) Model:

This model is used to designate shear strength as a function of depth. The depth is calculated from the top of the soil layer to the base center of a slice.

Basic

- ***Unit Weight*** Total unit weight of the soil.

- ***C-Top of Layer*** Undrained strength at the top of the soil layer.

- ***Rate of Increase*** Rate at which strength increases with depth.

- ***C - Maximum*** The maximum soil strength. If the Rate of Increase is negative, this value represents the minimum soil strength.

- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Advanced

- ***Unit Wt. Above WT*** Total unit weight of the soil above the water table (i.e., above the zero pressure line).

- ***Anisotropic Function*** A function of the modifier factor versus the base inclination angle of each slice. When this function is defined, the shear strength along the base is multiplied by the modifier factor obtained from the function. You can specify this function using the KeyIn Strength Functions: Anisotropic command.

- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

S=f(datum) Model:

This model is used to designate shear strength as a function of depth. The depth is calculated from a specified datum to the base center of a slice.

Basic

- ***Unit Weight*** Total unit weight of the soil.

- ***C-Datum*** Undrained strength at the top of the soil layer.

- ***Datum (elevation)*** Elevation (*y*-coordinate) of the datum line.

- ***Rate of Increase*** Rate at which strength increases with depth.

- ***C - Maximum*** The maximum soil strength. If the Rate of Increase is negative, this value represents the minimum soil strength.

- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Advanced

- ***Unit Wt. Above WT*** Total unit weight of the soil above the water table (i.e., above the zero pressure line).
- ***Anisotropic Function*** A function of the modifier factor versus the base inclination angle of each slice. When this function is defined, the shear strength along the base is multiplied by the modifier factor obtained from the function. You can specify this function using the KeyIn Strength Functions: Anisotropic command.
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Anisotropic Strength Model:

This model is used to designate anisotropic soil strength. Both vertical and horizontal *c* and Phi values are specified. The *c* and Phi values are first adjusted for anisotropy before they are used in the shear strength computation. The two equations for anisotropic adjustment of *c* and Phi are:

$$c = c_v \sin^2 a + c_h \cos^2 a$$

-- and --

$$f = f_v \sin^2 a + f_h \cos^2 a$$

where:

a = the inclination of the slice base.

Basic

- ***Unit Weight*** Total unit weight of the soil.
- ***C - Horizontal*** Cohesion component of the shear strength in horizontal direction.
- ***C - Vertical*** Cohesion component of the shear strength in vertical direction.
- ***Phi - Horizontal*** Friction angle of the soil in horizontal direction.
- ***Phi - Vertical*** Friction angle of the soil in vertical direction.
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Advanced

- ***Unit Wt. Above WT*** Total unit weight of the soil above the water table (i.e., above the zero pressure line).
- ***Unsaturated Phi B*** The rate of shear strength increase with a change in negative pore-water pressure. When f^b is zero, all negative pore-water pressures are set to zero. When f^b is nonzero, the effect of the negative pore-water pressures is included in the analysis.
- ***C-Phi Corr. Coef.*** The correlation coefficient between *c* and Phi when a probabilistic analysis is used. Its value ranges from -1.0 to 1.0. See the Theory chapter for more information on this parameter.
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Shear/Normal Function Model:

This model is used to specify a general curved relationship between shear strength and normal stress.

Basic

- ***Unit Weight*** Total unit weight of the soil.

- ***Function #*** The shear/normal function number. This function describes the shear strength of the soil as a function of normal stress. This function is specified by the KeyIn Strength Functions: Shear/Normal command.

- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Advanced

- ***Unit Wt. Above WT*** Total unit weight of the soil above the water table (i.e., above the zero pressure line).

- ***Anisotropic Function*** A function of the modifier factor versus the base inclination angle of each slice. When this function is defined, the shear strength along the base is multiplied by the modifier factor obtained from the function. You can specify this function using the KeyIn Strength Functions: Anisotropic command.

- ***Unsaturated Phi B*** The rate of shear strength increase with a change in negative pore-water pressure. When f^b is zero, all negative pore-water pressures are set to zero. When f^b is nonzero, the effect of the negative pore-water pressures is included in the analysis.

- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Anisotropic Function Model:

This is a general strength model for anisotropic soil. The variation of c and phi with respect to the base inclination angles is described by a general function. The input c and phi values are multiplied with the modifier factor obtained from the function before used in the shear strength computation.

Basic

- ***Unit Weight*** Total unit weight of the soil.

- ***Cohesion*** The cohesion component of the shear strength.

- ***Phi*** Friction angle of the soil.

- ***C - Anisotropic Function*** A function of the modifier factor versus the base inclination angle of each slice. When this function is defined, the specified Cohesion value is multiplied by the modifier factor obtained from the function. You can specify this function using the KeyIn Strength Functions: Anisotropic command.

- ***Phi - Anisotropic Function*** A function of the modifier factor versus the base inclination angle of each slice. When this function is defined, the specified Phi value is multiplied by the modifier factor obtained from the function. You can specify this function using the KeyIn Strength Functions: Anisotropic command.

- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Advanced

- ***Unit Wt. Above WT*** Total unit weight of the soil above the water table (i.e., above the zero pressure line).

- ***Unsaturated Phi B*** The rate of shear strength increase with a change in negative pore-water pressure. When f^b is zero, all negative pore-water pressures are set to zero. When f^b is nonzero, the effect of the negative pore-water pressures is included in the analysis.

- ***C-Phi Corr. Coef.*** The correlation coefficient between *c* and f when a probabilistic analysis is used. Its value ranges from -1.0 to 1.0. See the Theory chapter for more information on this parameter.

- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Combined, S=f(depth) Model:

With this model, the soil strength is based on c and f up to a maximum undrained strength C_u. Both c and C_u can vary with depth below the top of the soil layer.

Basic

- ***Unit Weight*** Total unit weight of the soil.
- ***Phi*** Friction angle of the soil.
- ***C - Top of Layer*** Cohesion at the top of the soil layer.
- ***C Rate Increase*** Rate at which cohesion increases with depth.
- ***Cu - Top of Layer*** Undrained strength, C_u, (cohesion) at the top of the soil layer.
- ***Cu Rate Increase*** Rate at which the undrained strength C_u increases with depth below the top of the layer.
- ***C / Cu Ratio*** The drained strength c is computed as a ratio of the undrained strength C_u when this ratio is not zero. When this ratio is zero, the drained strength c is computed from the C - Top of Layer value and the C Rate Increase value.
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Advanced

- ***Unit Wt. Above WT*** Total unit weight of the soil above the water table (i.e., above the zero pressure line).
- ***Anisotropic Function*** A function of the modifier factor versus the base inclination angle of each slice. When this function is defined, the shear strength along the base is multiplied by the modifier factor obtained from the function. You can specify this function using the KeyIn Strength Functions: Anisotropic command.
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Combined, S=f(datum) Model:

With this model, the soil strength is based on c and f up to a maximum undrained strength C_u. Both c and C_u can vary with depth below the datum reference position.

Basic

- ***Unit Weight*** Total unit weight of the soil.
- ***Phi*** Friction angle of the soil.
- ***C - Datum*** Cohesion at the datum reference position.
- ***C Rate Increase*** Rate at which cohesion increases with depth.
- ***Cu - Datum*** Undrained strength, C_u, (cohesion) at the datum reference position.
- ***Cu Rate Increase*** Rate at which the undrained strength C_u increases with depth below the datum reference position.
- ***C / Cu Ratio*** The drained strength c is computed as a ratio of the undrained strength C_u when this ratio is not zero. When this ratio is zero, the drained strength c is computed from the C-Datum value and the C Rate Increase value.

- ***Datum (elevation)*** Elevation (*y*-coordinate) of the datum reference position.
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Advanced

- ***Unit Wt. Above WT*** Total unit weight of the soil above the water table (i.e., above the zero pressure line).
- ***Anisotropic Function*** A function of the modifier factor versus the base inclination angle of each slice. When this function is defined, the shear strength along the base is multiplied by the modifier factor obtained from the function. You can specify this function using the KeyIn Strength Functions: Anisotropic command.
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

S=f(overburden) Model:

With this model, the soil strength is a function of the effective overburden stress above the base center of each slice. The effective overburden is computed from the weight of the slice and the pore water pressure acting on the base center. The shear strength is calculated as:

$$shear\ strength = effective\ overburden \times \frac{Tau}{Sigma\ Ratio}$$

Basic

- ***Unit Weight*** Total unit weight of the soil.
- ***Tau/Sigma Ratio*** A multiplication factor (e.g., 0.4 means that the shear strength is equal to 40% of the effective overburden).
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

Advanced

- ***Unit Wt. Above WT*** Total unit weight of the soil above the water table (i.e., above the zero pressure line).
- ***Anisotropic Function*** A function of the modifier factor versus the base inclination angle of each slice. When this function is defined, the shear strength along the base is multiplied by the modifier factor obtained from the function. You can specify this function using the KeyIn Strength Functions: Anisotropic command.
- ***S.D.*** The standard deviation of each parameter when a probabilistic analysis is used.

KeyIn Strength Functions Shear/Normal

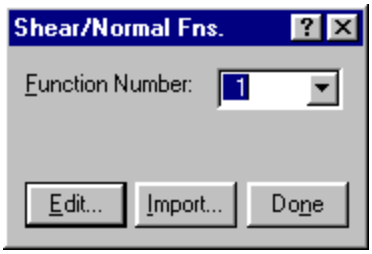
☐ Defines the relationship between shear stress and normal stress for the soil properties.

A shear/normal strength function describes the shear and normal stress relationship. This function is useful for implementing a curved, nonlinear failure envelope that can be applied to the soil properties using the KeyIn Soil Properties command.

Defining Each Function Data Point

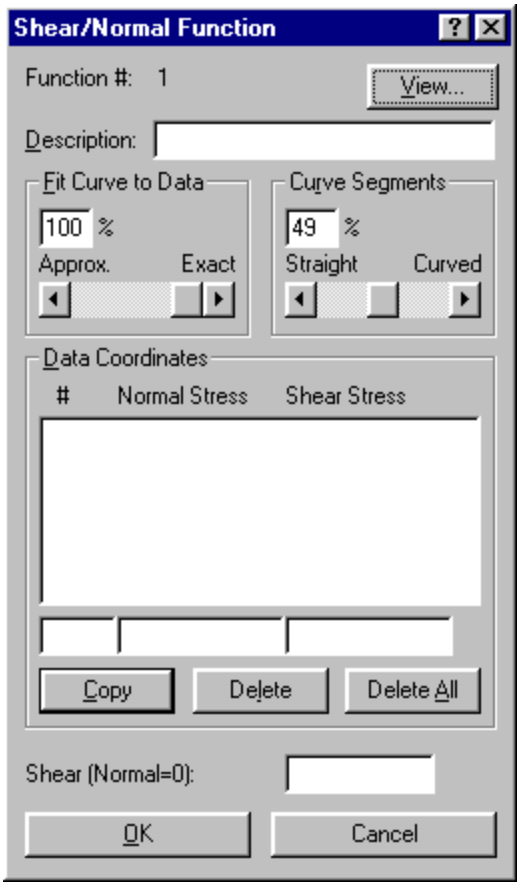
➢ **To define each data point in a shear/normal strength function:**

1. Choose KeyIn Strength Functions Shear/Normal. The following dialog box appears:



2. In the Function Number edit box, type the function number to define.

3. Select Edit. The following dialog box appears to let you enter the data points in the function:



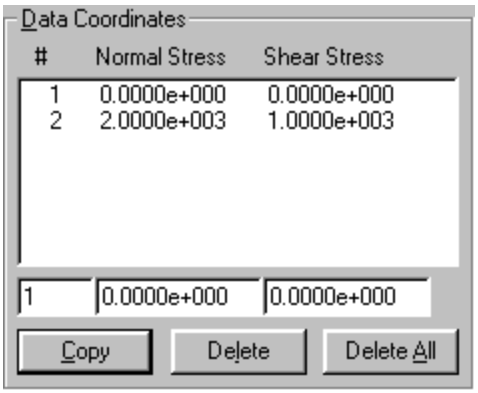
Steps 4 to 7 define the extremities of the function, allowing you to later use the Graph window to visually define the function points.

4. Enter the minimum *x*- and *y*-coordinates by typing **1** in the # edit box, the minimum normal stress value in the Normal Stress edit box, and the minimum shear stress in the Shear Stress edit box.

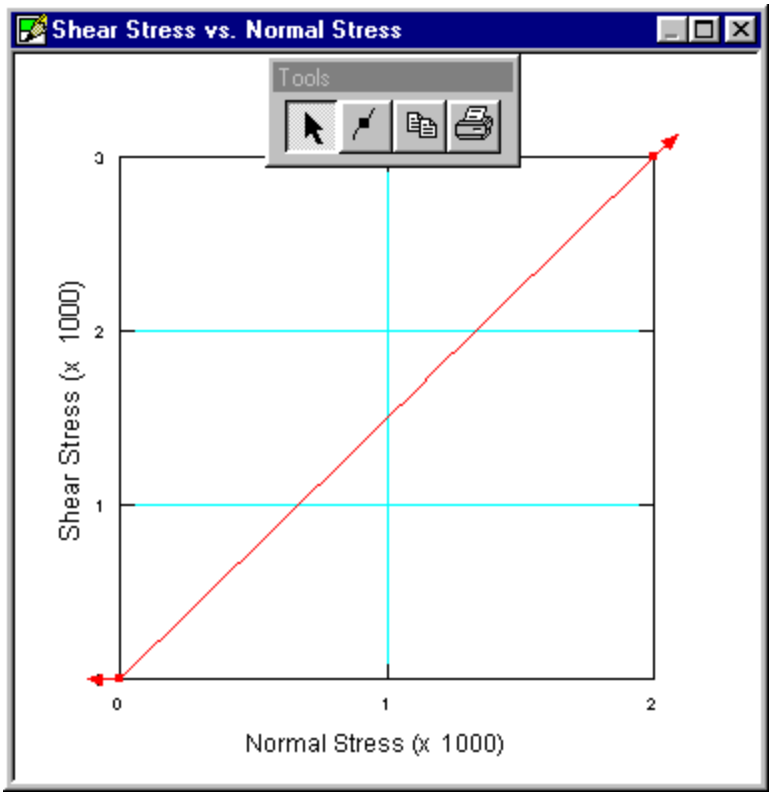
5. Select Copy. The values in the edit boxes are copied into the list box.

6. Enter the maximum *x*- and *y*-coordinates by typing **2** in the # edit box, the maximum normal stress value in the Normal Stress edit box, and the maximum shear stress in the Shear Stress edit box.

7. Select Copy. The values in the edit boxes are copied into the list box. The following list box contains two typical points:



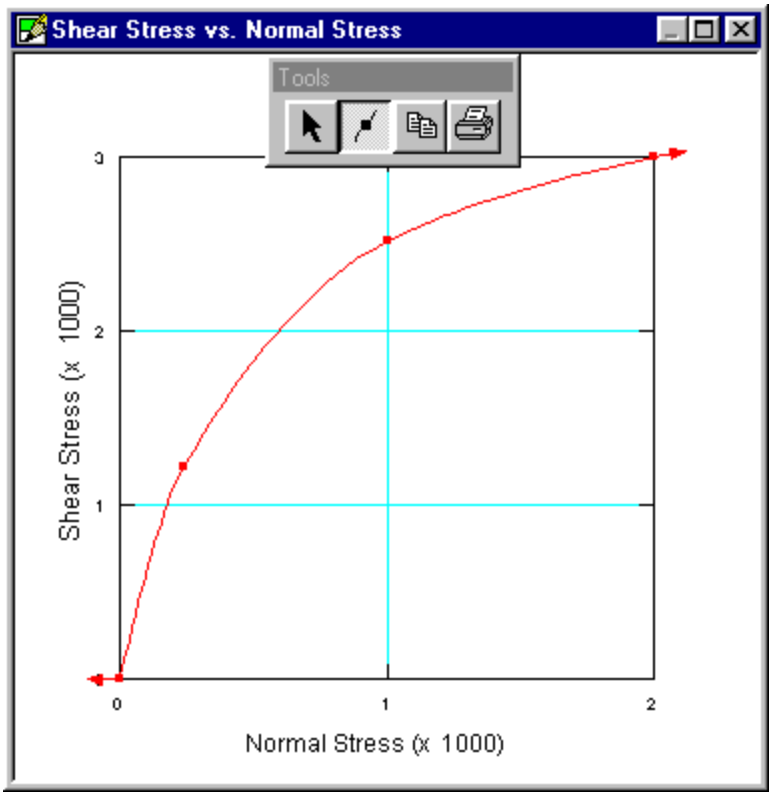
8. Once the function extremities have been entered, select View to display the function graph.



When the View button is pressed, SLOPE/W computes a graph scale encompassing the function extremities and a spline function through the data points. The arrows at the end of the data points represent how SLOPE/W interprets the function beyond the extremities.

9. Use the tools in the Graph tool box to complete the shear/normal strength function definition. The Graph tool box allows you to add, move, and delete points interactively. You can also adjust the curvature of the spline between data points and the degree to which the spline is fit to the data points. These features are discussed later in this section in more detail.

The following graph shows a typical shear/normal strength function:



10. Type an appropriate name for the function in the Description edit box. The function name is helpful when deciding which function to edit or import.

11. Double-click on the control-menu box to close the View window.

12. Select OK. The initial KeyIn Functions dialog box appears.

13. Select Done to exit this command.

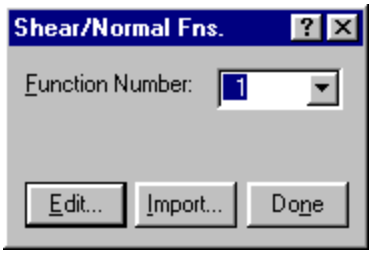
    Or, type a new function number and select OK to define another shear/normal strength function.

Importing and Modifying a Shear/Normal strength function

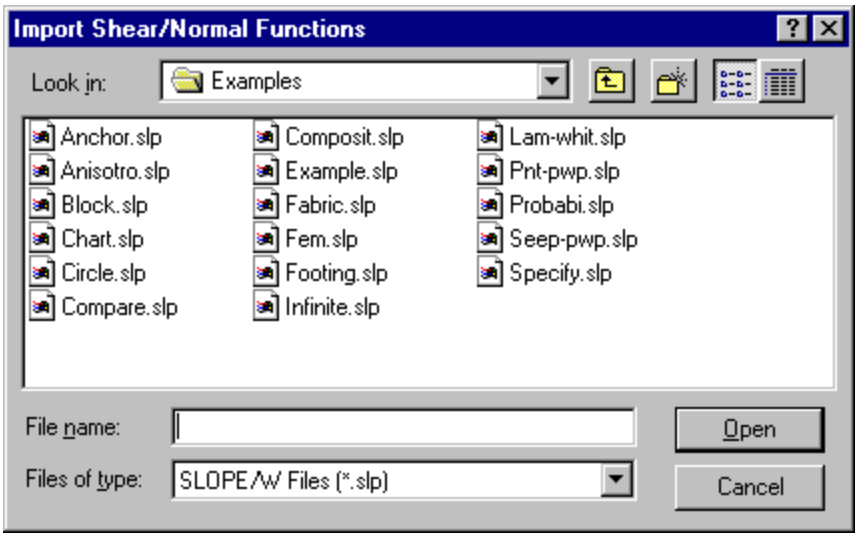
It may be convenient to define a shear/normal strength function by modifying an existing function. SLOPE/W allows you to import a function from another problem. The imported function can then be modified to suit the current problem.

➢ **To import a function into the current problem:**

1. Choose KeyIn Strength Functions Shear/Normal. The following dialog box appears:

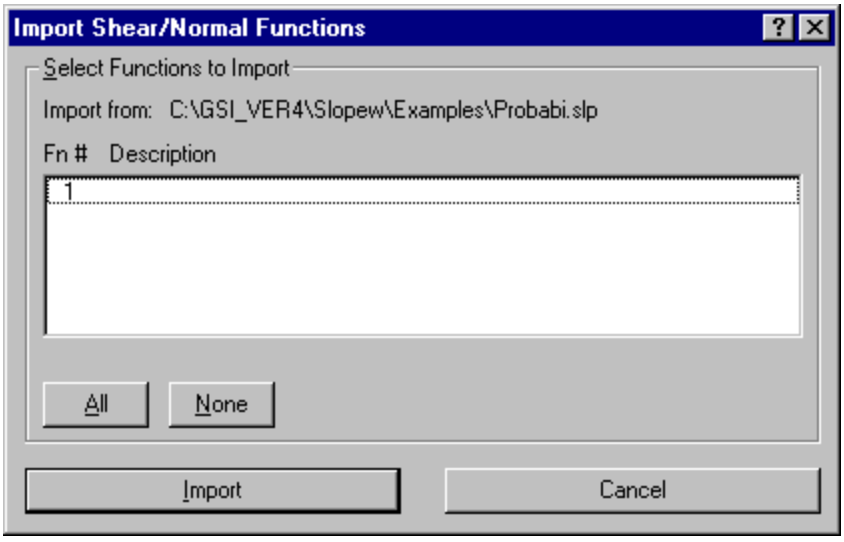


2. To import a function from an existing problem, select the Import button. The following dialog box appears:



3. Select the problem data file that contains the shear/normal strength function to import.

4. Select OK in the Import Shear/Normal Functions dialog box. The following dialog box appears to enable you to select the functions to import:



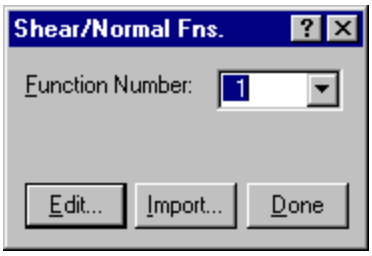
5. In the dialog list box, select the functions to import. Select All to select all functions or None to remove the selection from all functions. You can also click on functions individually. A group of functions can be selected either by pressing the CTRL key and clicking on each function in the group or by pressing the SHIFT key and clicking on the first and last function in the group.

6. Select Import to import the selected shear/normal strength functions into the current problem.

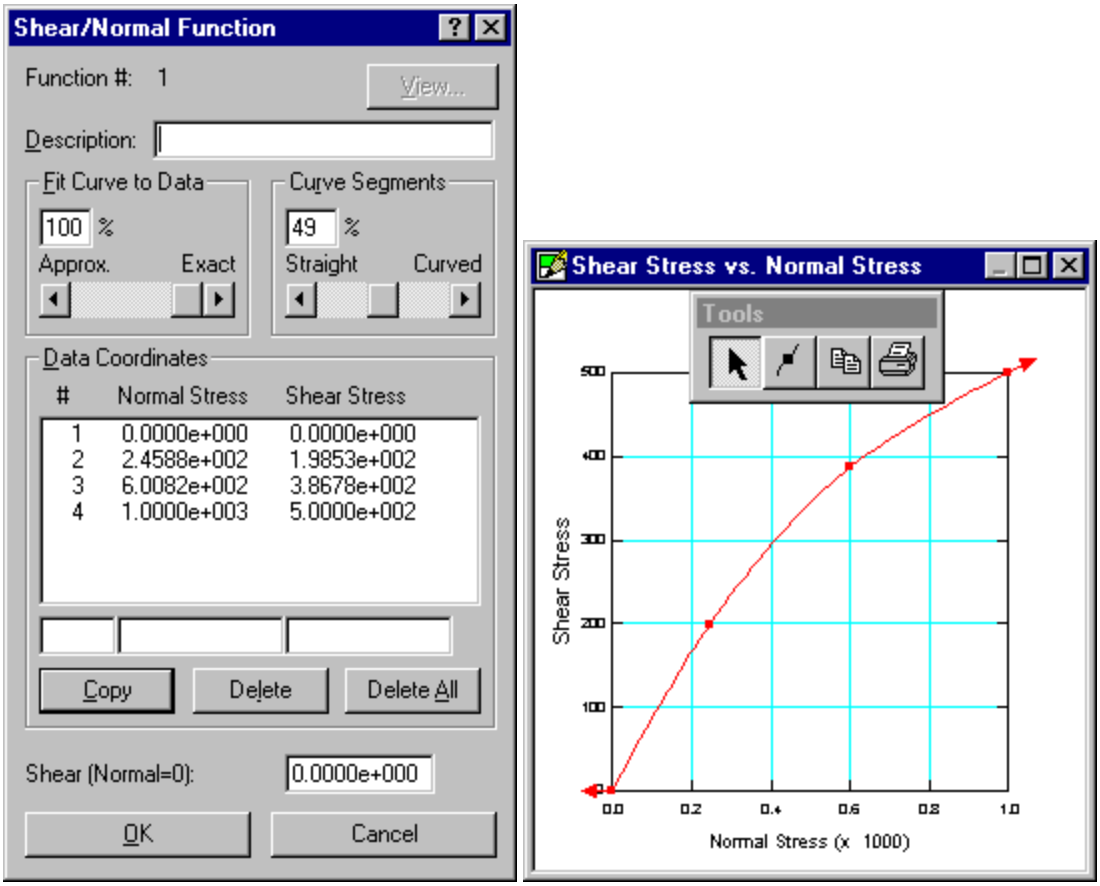
   The imported functions are added to the end of the list of existing shear/normal strength functions in the Shear/Normal Functions edit box, and the first imported function number is selected in the Function Number edit box. Select Edit to modify the function.

➢ **To modify an existing shear/normal strength function:**

1. Choose KeyIn Strength Functions Shear/Normal and select the function number to edit in the Function Number drop-down list box:



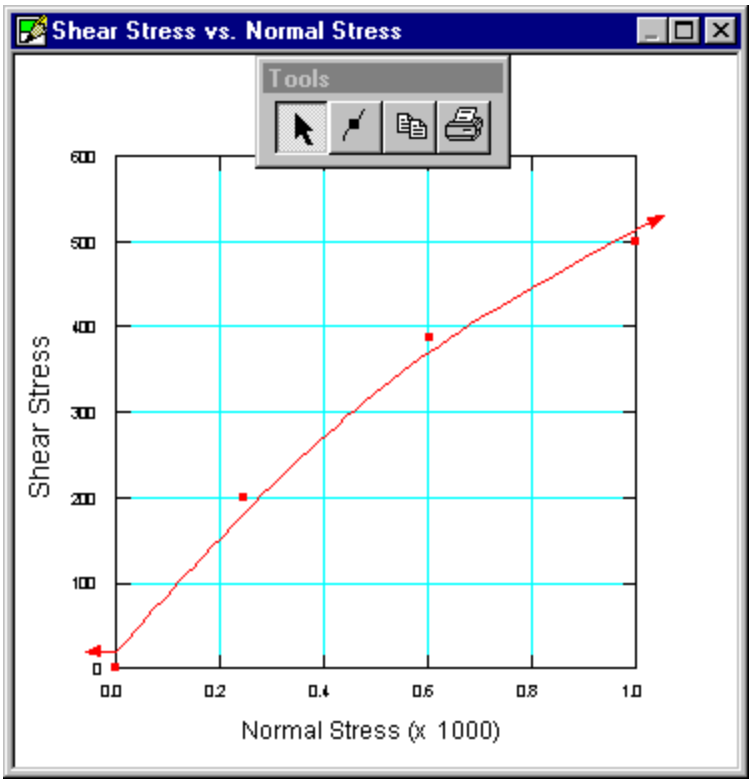
2. Select Edit. The Edit Shear/Normal strength function dialog box appears, along with the Graph window, to let you modify the data points in the function:



3. To move the function up or down, type a new shear stress value in the Shear (Normal=0) edit box, and press the TAB key. The function data points are moved up or down to reflect the new shear stress value at zero normal stress.

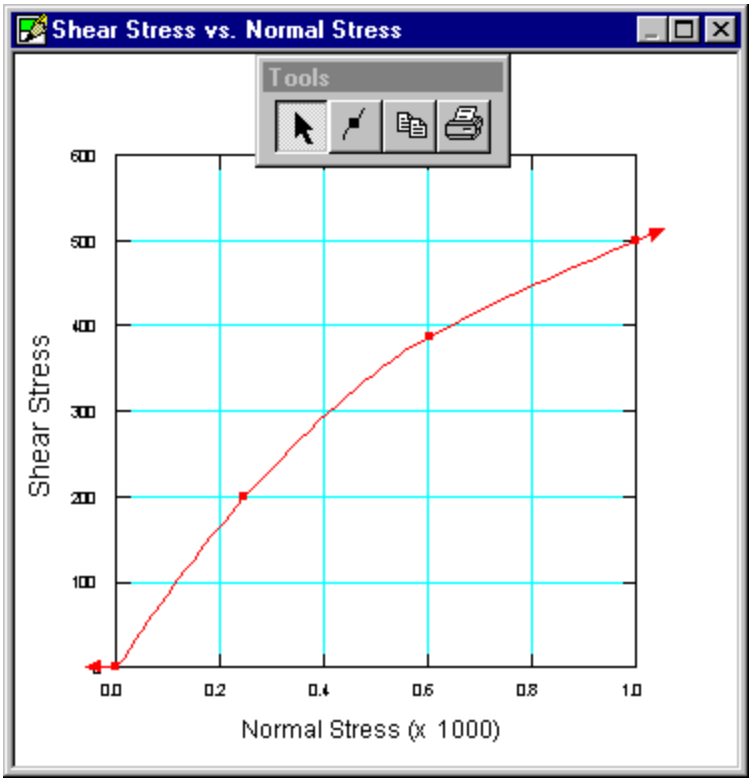
4. To fit the curve more or less exactly to the data points, specify a new value in the Fit Curve to Data group box either by moving the scroll bar or typing a percentage value. When the curve is fit exactly (100%) to the data points, the spline passes through each data point. As the curve fitting is reduced, the spline shape approaches a straight line that passes close to each data point. This is useful when you want to approximate a spline through laboratory-measured data points without moving any of the data points.

   The following spline curve is fit to the data using a value of 30%:



5. To change the shape of the spline curve between data points, specify a new value in the Curve Segments group box either by moving the scroll bar or typing a percentage value. When the curve segments are curved (100%) between data points, the curve is defined as a natural spline. As the curve segments are made straighter, the curve segments approach a straight line between data points. Straightening the curve segments helps to prevent "spline overshoot" (extreme peaks or valleys in the spline). It also allows you to define "step" functions that have straight line segments between each data point.

The following spline curve uses a curvature value of 5%:

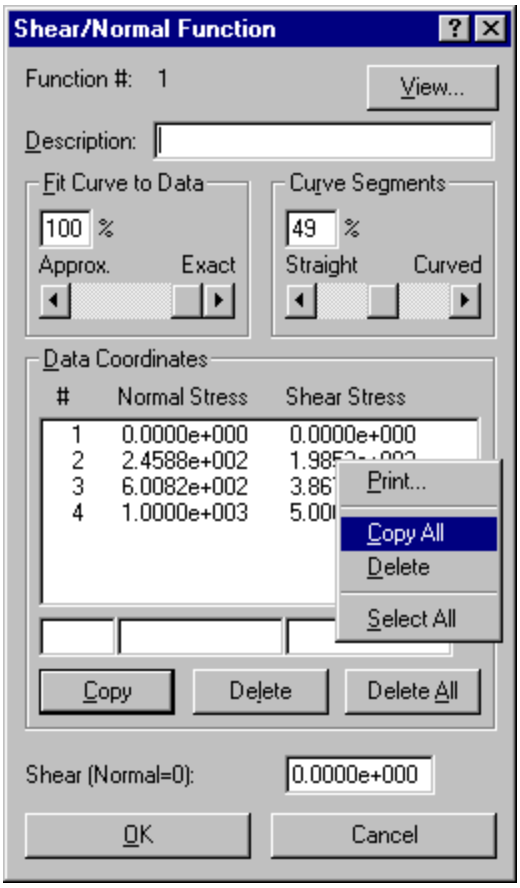


Copying Function Data Points to the Clipboard

SLOPE/W allows you to copy the function data points to the Windows clipboard so that they can be accessed by other Windows applications, such as Microsoft Word or Excel.

➢ **To copy the function data points to the Windows Clipboard:**

1. In the Edit Shear/Normal strength function dialog box, click the right mouse button to display a context-sensitive pop-up menu:

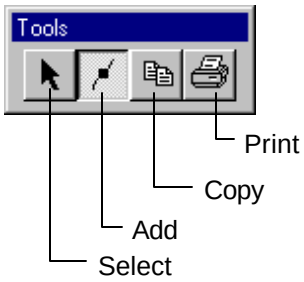


2. Select Copy All from the pop-up menu to copy all data coordinates into the clipboard.

More function editing techniques are discussed in the following section that describes the Graph tool box.

The Graph Window Toolbar

The Graph window toolbar contains buttons for moving and deleting points, adding points, copying the graph to the Clipboard, and printing the graph.



To access a command from the toolbar, click the button with the left mouse button. Clicking on the Select button puts you in Select mode, while clicking on the Add button puts you in Add mode. The Copy and Print buttons can be used while you are in either mode. The toolbar commands are:

Select Select Mode allows you to select one or more function points for moving or deleting. This is the default mode for the Graph window.

- To select a point, click the left mouse button near the point. To select a group of points, drag a rectangle around the points.
- Once points are selected, they can be deleted by pressing the DELETE key. They can be moved by clicking on one of the selected points and holding the left mouse button down, dragging the mouse to a new position, and then releasing the left mouse button. Alternatively, you can move the points with the arrow keys. Whenever points are moved, SLOPE/W recalculates the spline curve between the function data points.
- Data points can also be selected in the dialog list box either by pressing the CTRL key and clicking on each point in the group or by pressing the SHIFT key and clicking on the first and last point in the group.

Add Add Mode allows you to add a function point to the graph.

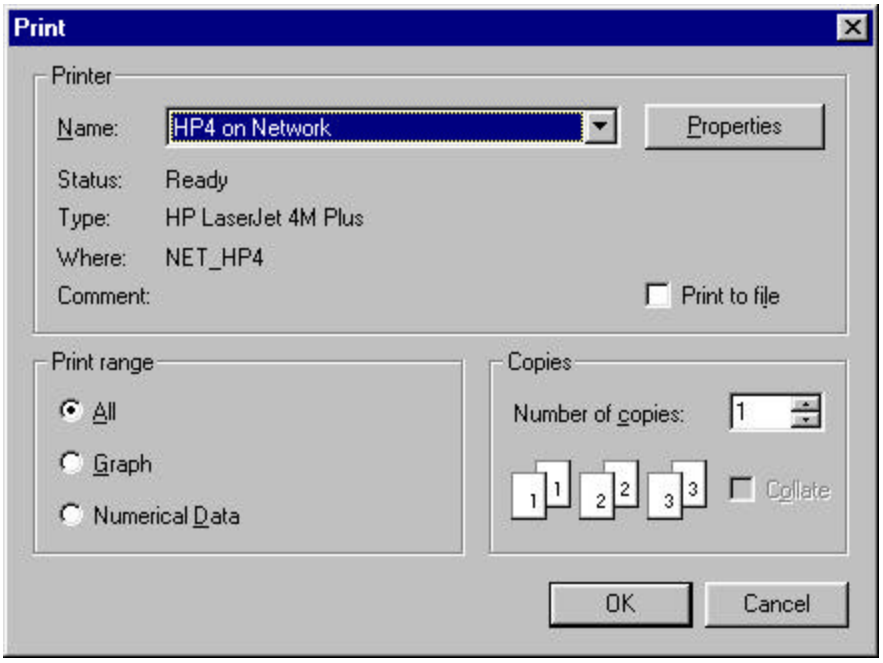
- To add a point, click the left mouse button at the desired position. SLOPE/W adds the point to the graph and recalculates the spline curve between the function data points.

Copy Copies the graph to the Clipboard.

- This button allows you to transfer the graph to another Windows application for creating reports, slide presentations, or enhancing the graph. A beep is sounded when the graph has been copied to the Clipboard. To display the contents of the Clipboard, run the Windows Clipboard Viewer program.

Print Prints the graph on the printer.

- Select the Print button to print the graph. The following dialog box appears:



- Select the printer from the Printer Name drop-down list box. If you wish to change the printer settings, select the Properties button.
- Select either the All, Graph, or Numerical Data from the Print range Options. If you select All, both the Graph and the Numerical Data (i.e., the function data point coordinates) will be printed

- Select OK to print the graph and/or data.

  The graph is printed on the default printer at the size it is displayed on screen. Resizing the Graph window changes the printed size of the graph. If the graph is larger than the printer page size, the graph will be printed at the printer page size.

Whenever a point is selected, moved, deleted, or added in the Graph window, the dialog list box is updated to reflect the change. Likewise, when a point is modified in the dialog list box, the Graph window is also updated. This feature allows you to switch between the KeyIn Functions dialog box and the Graph window while you are defining the function.

The points are sorted by their *x*-coordinates whenever points are moved, added, or deleted from the graph or from the dialog box. This feature allows you to move the points anywhere on the graph without destroying the function.

Comments

Function numbers should be specified in a continuous series. For example, if you are defining three functions, assign them function numbers of 1, 2, and 3. While you may choose any integer as a function number, large integers will decrease the efficiency of SLOPE/W.

The first data point must always be (0,0). The remaining normal and shear stress values must be positive.

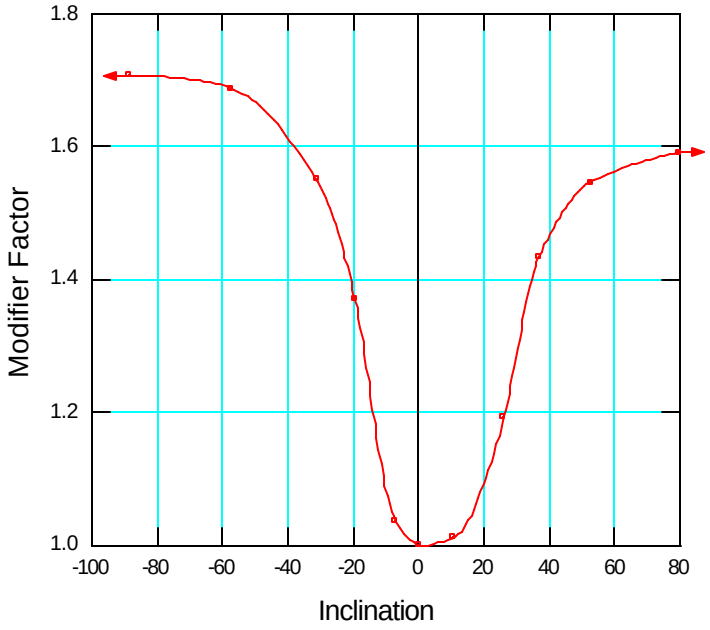
When specifying normal stress values, note the following:

- No two normal stress values should be the same.
- The normal stress values must be in ascending order. SLOPE/W sorts the function points by the normal stress values when you select View or OK.
- The slope (gradient) of the shear/normal strength function must always be positive over its entire range.
- A straight line function can be defined by specifying only two data points.
- The Graph window can be resized to create a different size of graph or maximized to create the largest possible graph. When the window is enlarged horizontally, the graph appears to be flatter. This is because the *x*- and *y*-axes are always scaled to fit the entire window area; resizing the window does not affect the point coordinates.
- The font used in the Graph window can be changed by using the View Preferences command.

KeyIn Strength Functions Anisotropic

☐ Defines the relationship between soil strength and the slice inclination angle.

The anisotropic strength function can be used to describe a general relationship between the angle of inclination at the base of the slice and the soil strength. Anisotropic strength functions are useful where site-specific data is available or in cases where the soil stratification is inclined. The following is an example of a typical anisotropic strength function:



Anisotropic strength functions are defined and modified in the same way as shear/normal strength functions. See the KeyIn Strength Functions: Shear/Normal section for more information on defining and modifying functions.

Comments

Function numbers should be specified in a continuous series. For example, if you are defining three functions, assign them function numbers of 1, 2, and 3. While you may choose any integer as a function number, large integers will decrease the efficiency of SLOPE/W.

When specifying inclination values, note the following:

- No two inclination values should be the same.
- Inclination angles must be between -90° and +90°.
- The inclination values must be in ascending order. SLOPE/W sorts the function points by the inclination values when you select View or OK.
- Positive inclination angles are for slices moving down-slope (usually near the slope crest) and negative angles are for slices moving up-slope (often in the toe area).
- A straight line function can be defined by specifying only two data points.

- The Graph window can be resized to create a different size of graph or maximized to create the largest possible graph. When the window is enlarged horizontally, the graph appears to be flatter. This is because the *x*- and *y*-axes are always scaled to fit the entire window area; resizing the window does not affect the point coordinates.

- The font used in the Graph window can be changed by using the View Preferences command.

KeyIn Tension Crack

☐ Defines the tension crack line or angle.

This command allows you to define the tension crack line or tension crack angle, depending on the tension crack option selected with KeyIn Analysis Control.

If the Tension Crack Angle option was selected using KeyIn Analysis Control, you can model a tension crack without having to specify its depth. The tension crack angle controls the depth by limiting the inclination of the slip surface. When the inclination of a line tangent to a circular slip surface becomes steeper than the specified value, the slip surface is projected vertically to the ground surface to simulate a tension crack. Figure 4.10 illustrates the tension crack angle.

The following dialog box appears when you choose KeyIn Tension Crack:

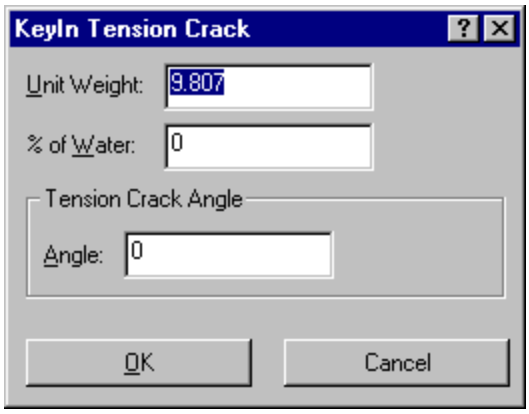
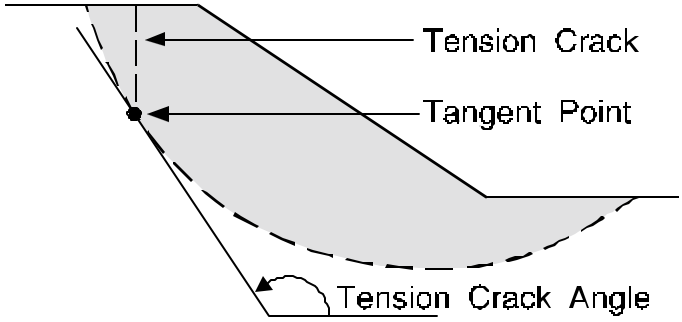
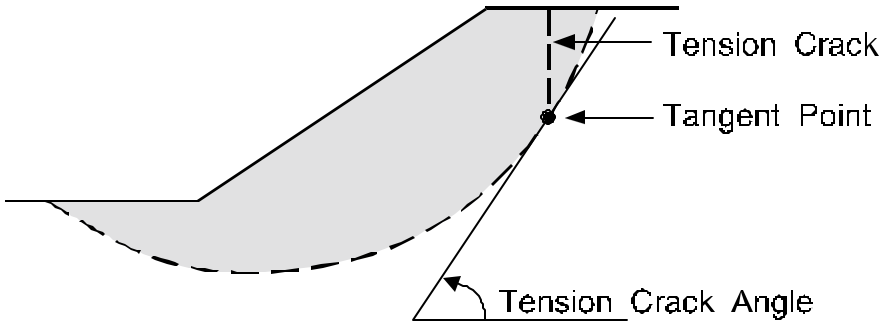


Figure 4.10 Definition of a Tension Crack Angle

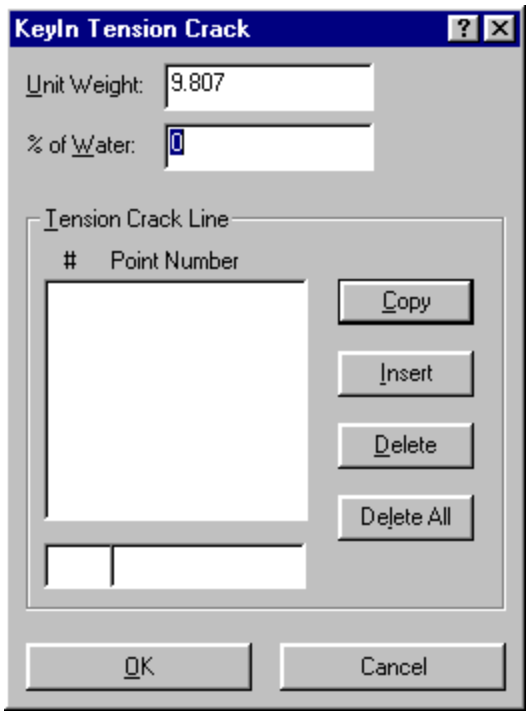
(a) Left-to-Right Problem



(b) Right-to-Left Problem



If the Tension Crack Line option was selected using KeyIn Analysis Control, the tension crack is modelled by specifying a line across the geometry. The slip surface is projected vertically to the top soil surface at the point where the slip surface intersects the tension crack line. The following dialog box appears when you choose KeyIn Tension Crack:



Unit Weight Defines the unit weight of a fluid that fills a tension crack.

% of Water Specifies the percentage of fluid (from 0 to 1) in the tension crack. A value of 0.0 represents a dry tension crack, while a value of 1.0 represents a completely water-filled tension crack.

Tension Crack Angle Specifies the tangent angle along a circular slip surface.

Tension Crack Line Specifies the points in the tension crack line.

The primary method of defining the points in the tension crack line is by drawing them on the screen with Draw Tension Crack Line. The main purpose of KeyIn Tension Crack is to:

- Modify the point numbers in a line.
- Delete points from a line.

➢ **To define or modify the tension crack line points:**

1. In the # edit box, type a number indicating where the point will be added on the line (i.e., 1 will add the point to the beginning of the line).
2. In the Point Number edit box, type the point number to add to the line.
3. Select Copy to transfer the point data to the list box.
4. Repeat Steps 1 to 3 for all points in the line to define.
5. To modify a point in the line, click on the point in the list box with the left mouse button.

   The position of the point in the line and the point number are copied into the edit boxes.
6. Delete the point by selecting Delete or modify it by typing in a new point number and selecting Copy.
7. To select multiple points in the list box for deletion, either press the CTRL key and click on each point to delete or press the SHIFT key and click on the first and last point to delete. Click on the Delete button to delete all selected points.

➢ **To delete all points in the tension crack line:**

- Select Delete All.

To delete or move the tension crack line points graphically, choose the Modify Objects command. Deleting one of the points defining the tension crack line will remove the line. Moving a tension crack line point will also move the line.

Comments

The tension crack line must start at the left extremity of the problem and extend across the problem to the right extremity. In other words, the starting and ending *x*-coordinates of the line must match the left and right boundaries of the problem.

Choose the Tools Verify command to help you verify that the tension crack line has been defined correctly.

The tension crack line must be defined from left to right. In other words, the *x*-coordinate of each point in the line must be greater than the *x*-coordinate of the previous point in the line. Vertical segments in the line are not permitted.

The tension crack line cannot lie above the top soil surface or within a soil that uses the No Strength soil model (that is, c=0 and f =0).

When the unit weight is greater than zero, SLOPE/W applies a hydrostatic horizontal force on the side of the tension crack. The magnitude of the hydrostatic force is defined as,

$$F = \frac{g\, d^2}{2} \qquad (4.2)$$

where:

g = the unit weight of the fluid in the tension crack

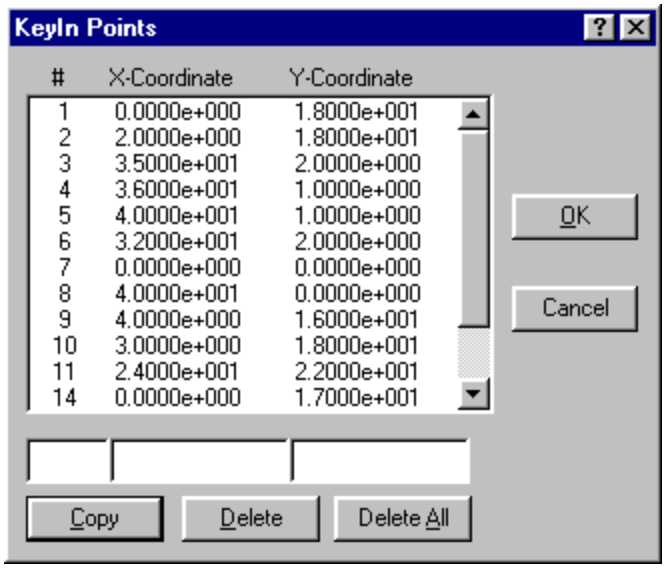
d = the depth of the tension crack

The tension crack force is applied at one-third of the depth from the bottom of the crack.

KeyIn Points

☐ Defines points used in specifying the geometric data.

When you choose KeyIn Points, the following dialog box appears:



Points are used to specify line segment endpoints, grid centers, line and anchor load positions, and all other data required by SLOPE/W. Since points are automatically created when using the DEFINE Draw commands, it normally is unnecessary to explicitly create points by choosing KeyIn Points or Draw Points.

The main purpose of KeyIn Points is to:

- View the point coordinates numerically.
- Refine the coordinates after they have been drawn.
- Delete point data.

➢ **To edit point data in the dialog box:**

1. Select the point to edit by clicking on the point in the list box with the left mouse button.

   The point number and the *x*- and *y*-coordinates are copied into the edit boxes.

2. Change the *x*- or *y*-coordinates by entering new values in the edit boxes. To create a new point, type in a point number that does not already exist.

3. Select Copy to transfer the data to the point in the list box that matches the point number in the edit box.

   If the point number does not already exist in the list box, a new point is created in the list box.

4. Repeat Steps 1 to 3 for all points to edit.

5. Select OK.

Points can be deleted by clicking on the point in the list box and selecting Delete. To delete all the points in the list box, select Delete All. To select multiple points in the list box for deletion, either press the CTRL key and click on each point to delete or press the SHIFT key and click on the first and last point to delete. Click on the Delete button to delete all selected points.

Comments

Deleting a point will not delete any lines that may be connected to the point; the point simply will be removed from the line data.

To delete or move points graphically, choose the Modify Objects command. Moving points will also move all objects attached to the points, such as soil lines.

To display the information defined at a specific point, choose the View Point Information command.

Figure 4.11 and Figure 4.12 show how points are used to define the geometry, grid of centers, and radius lines.

Figure 4.11 Designation of Points for the Geometry

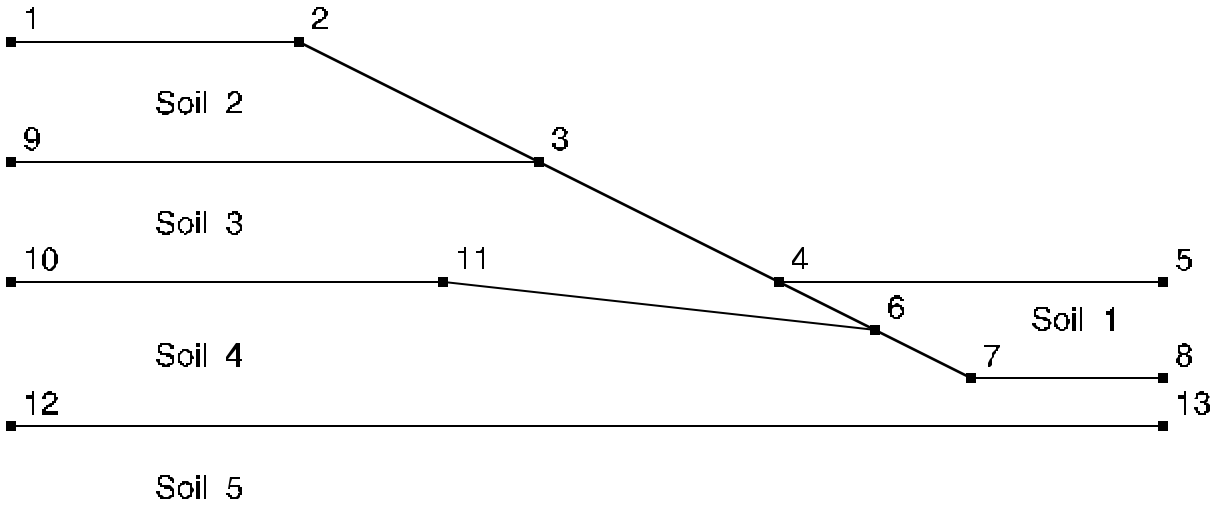
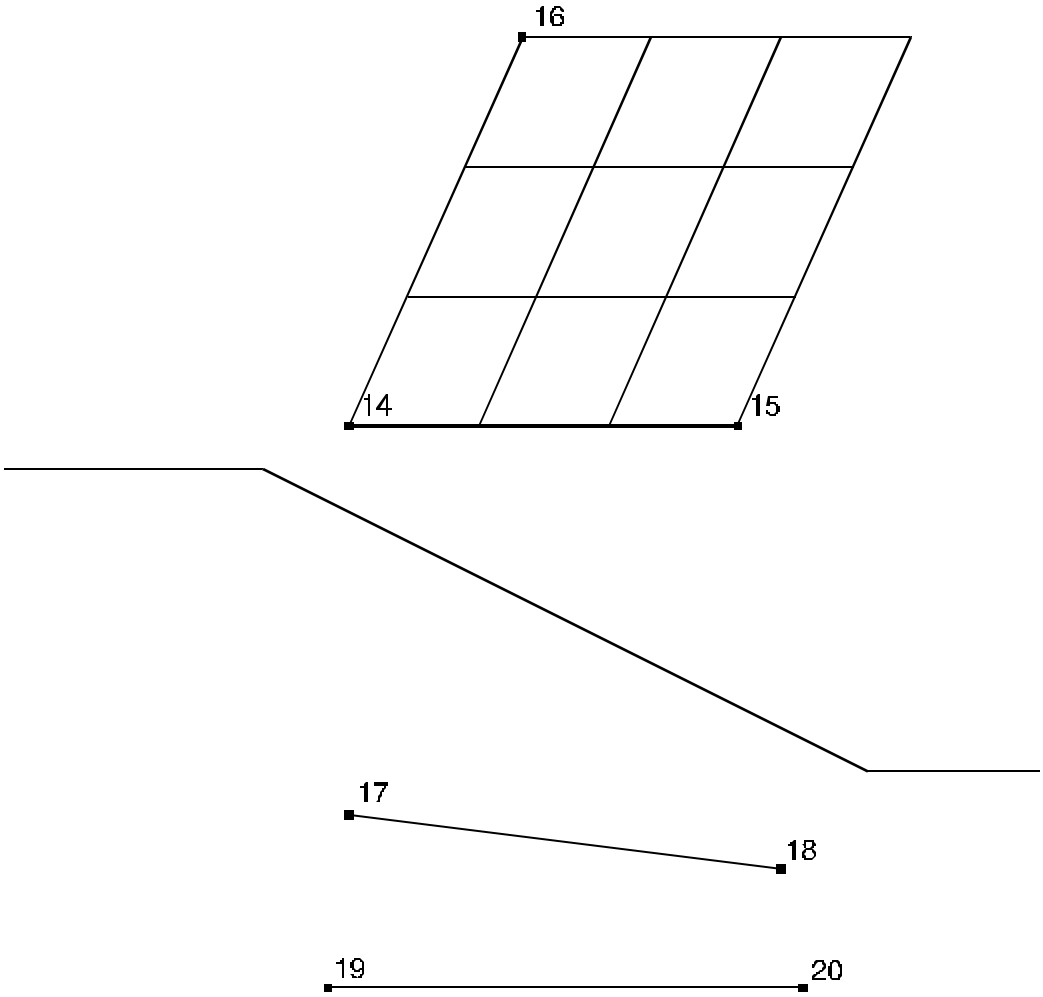


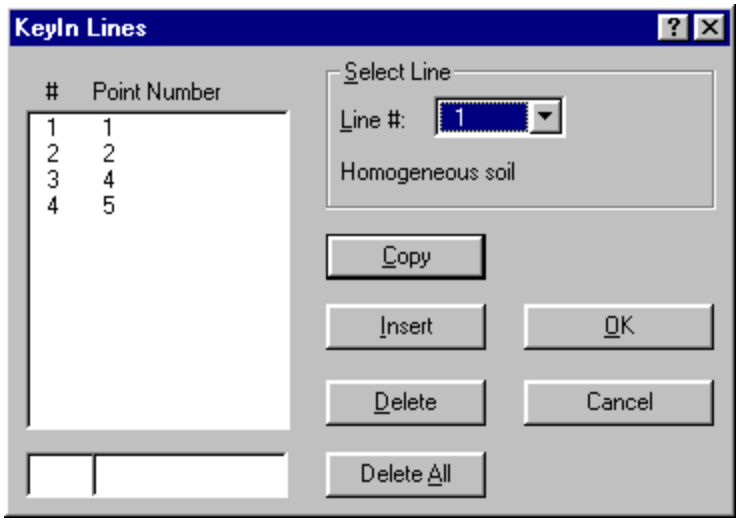
Figure 4.12 Designation of Points for the Grid of Centers and Radius Lines



KeyIn Lines

Defines the boundaries for each soil.

When you choose KeyIn Lines, the following dialog box appears:



The primary method of defining lines is by drawing them on the screen with Draw Lines.

The main purpose of KeyIn Lines is to:

- Check the data for individual lines.
- Modify the line data.
- Delete lines.

➢ **To define or modify line data in the dialog box:**

1. In the Select Line group box, select the line number from the drop-down list box. The list box contains one line number for each soil defined. If the selected line already contains points, the list box will be filled with the points contained in the line.
2. In the # edit box, type a number indicating where the point will be added on the line (i.e., 1 will add the point to the beginning of the line).
3. In the Point Number edit box, type the point number to add to the line.
4. Select Copy to transfer the point data to the list box.
5. Repeat Steps 2 to 4 for all points in the line to define.
6. To modify a point in the line, click on the point in the list box with the left mouse button.

   The position of the point in the line and the point number are copied into the edit boxes.
7. Delete the point by selecting Delete or modify it by typing in a new point number and selecting Copy.
8. Repeat Steps 1 to 7 for all lines to define.
9. Select OK.

➢ **To insert a new point in the line:**

1. Type a number in the # edit box to indicate where the point will be inserted in the line.

2. In the Point Number edit box, type the point number to insert in the line.

3. Select Insert.

   The new point is inserted in the list box. For example, if you insert a new point at the third position in a line, the point is inserted between the second and third points in the line.

Points can also be inserted graphically by using the Draw Lines command and clicking at the position on the line where you want the point to be inserted.

To delete all points in a line, select Delete All. To select multiple points in the list box for deletion, either press the CTRL key and click on each point to delete or press the SHIFT key and click on the first and last point to delete; then press the Delete button to delete the selected points.

Points can be deleted graphically by choosing Delete Points and clicking on the point to delete; the point is then removed from the line data.

Comments

Each line defines the top boundary of the corresponding soil number. The bottom boundary of the soil is defined by the subsequent line number. Figure 4.13 shows how the line and soil numbers are related.

Each line must start at the left extremity of the problem and extend across the problem to the right extremity. In other words, the starting and ending *x*-coordinates of each line must match the left and right boundaries of the problem.

The geometry lines must be specified in descending order, starting with the ground or water surface and ending with the bottom soil layer. Water impounded against a slope must be defined as Soil 1 (Line 1) or as a surface pressure line.

Lines must be defined from left to right. In other words, the *x*-coordinate of each point in a line must be greater than the *x*-coordinate of the previous point in the line. Vertical segments in a line are not permitted.

The geometry must be wide enough to include all potential slip surfaces; SOLVE will not compute a factor of safety for any slip surfaces that extend beyond the geometry. When a grid of slip surface centers is used, the left and right boundaries of the geometry should extend beyond the slip circle with the largest radius.

Lines defining discontinuous strata must also extend from the left to the right boundaries of the geometry. Beyond the point at which the strata becomes discontinuous, the top and bottom lines of the layer are superimposed, as shown in Figure 4.14.

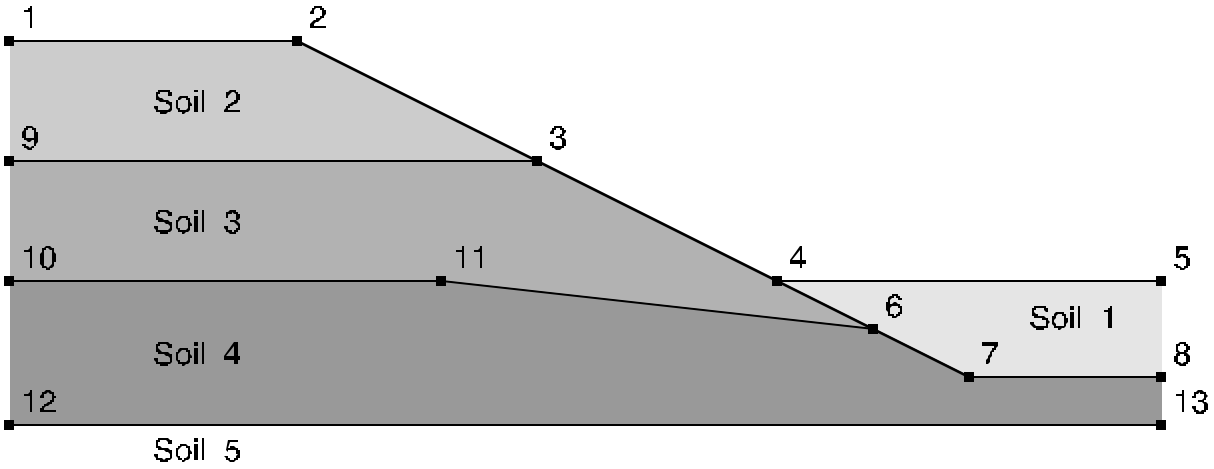
Lines must not cross other lines. Two lines can meet and overlap, but they cannot cross.

Choose the Tools Verify command to help you verify that the soil lines have been defined correctly.

To move or delete the points on a line, choose the Modify Objects command.

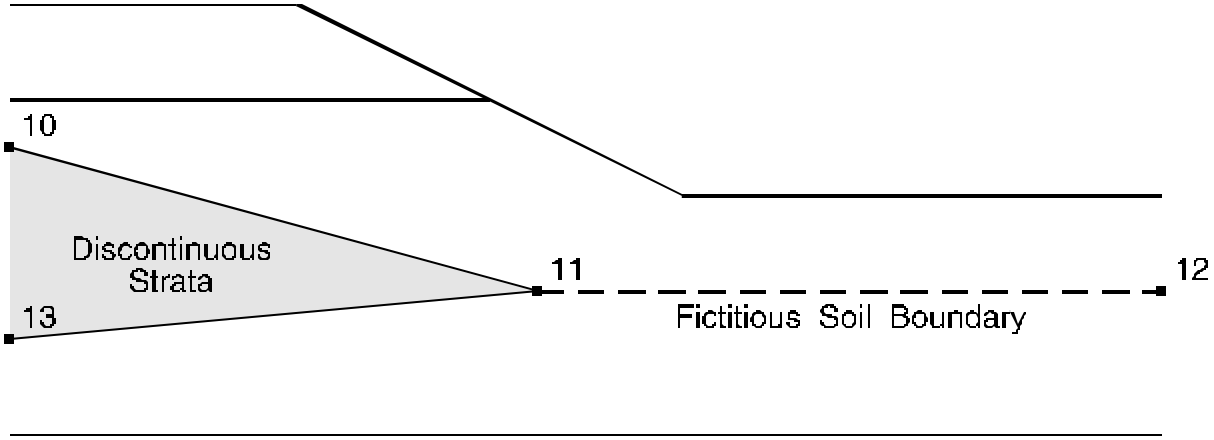
To graphically highlight all the points in a soil line, choose the View Soil Properties command.

Figure 4.13 Definition of Lines



Line 1: Points 1, 2, 3, 4, 5
Line 2: Points 1, 2, 3, 4, 6, 7, 8
Line 3: Points 9, 3, 4, 6, 7, 8
Line 4: Points 10, 11, 6, 7, 8
Line 5: Points 12, 13

Figure 4.14 Definition of a Discontinuous Strata

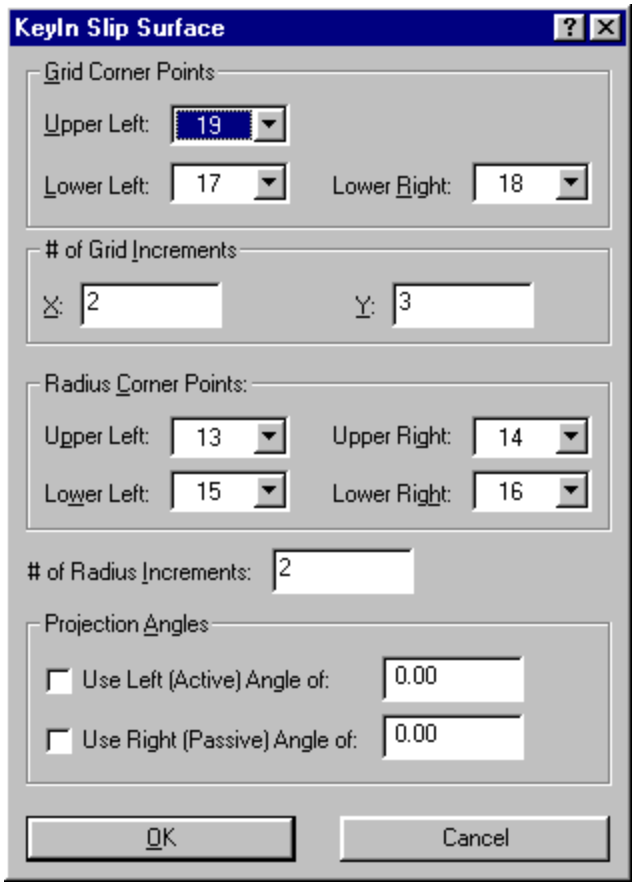


Top Line: Points 10, 11, 12
Bottom Line: Points 13, 11, 12

KeyIn Slip Surface Grid & Radius

☐ Defines the rotation centers and radii for circular and composite slip surfaces.

When you choose KeyIn Slip Surface Grid & Radius, the following dialog box appears:



The grid points are used as the center point of each circular slip surface. The radii of potential slip circles are defined by lines that are tangent to the circles. SLOPE/W computes the radius for each slip circle as the perpendicular distance from each "radius line" to each grid center.

Projection angles can be specified to limit the slip surface inclination in both the crest and toe areas of the slip surface.

The primary method of defining the grid and radius lines is by drawing them on the screen with the Draw Slip Surface: Grid and Draw Slip Surface: Radius commands.

The main purpose of KeyIn Slip Surface Grid & Radius is to:

- Specify slip surface projection angles.
- View the points and increments numerically.
- Edit the point numbers and increments.

➢ **To modify the grid, radius lines, and projection angles in the dialog box:**

1. To change any of the points in the Grid Corner Points group box, click on the down-arrow to the right of one of the corner point edit boxes, and select the desired point number.

2. Repeat Step 1 for all grid corner points to modify. Make sure that the points selected represent upper-left, lower-left, and lower-right grid corner points respectively.

3. Type the *x* and *y* grid increments in the # of Grid Increments X and Y edit boxes.

4. To change any of the points in the Radius Corner Points group box, click on the down-arrow to the right of one of the corner point edit boxes, and select the desired point number.

5. Repeat Step 4 for all radius corner points to modify. Make sure that the points selected represent upper-left, upper-right, lower-left, and lower-right grid corner points respectively.

6. Type the number of radius increments in the # of Radius Increments edit box.

7. To specify a projection angle along the left side of the slip surface, check the Left Angle check box in the Projection Angles group box and specify an angle in the edit box.

   If the left projection angle is on the active (crest) side of the problem (i.e., the slip surface movement is from left to right), the angle must be between 100º and 135º; if the left projection angle is on the passive (toe) side, it must be between 120º and 180º.

8. To specify a projection angle along the right side of the slip surface, check the Right Angle check box in the Projection Angles group box and specify an angle in the edit box.

   If the right projection angle is on the active (crest) side of the problem (i.e., the slip surface movement is from right to left), the angle must be between 45º and 80º; if the right projection angle is on the passive (toe) side, it must be between 0º and 60º.

9. Select OK.

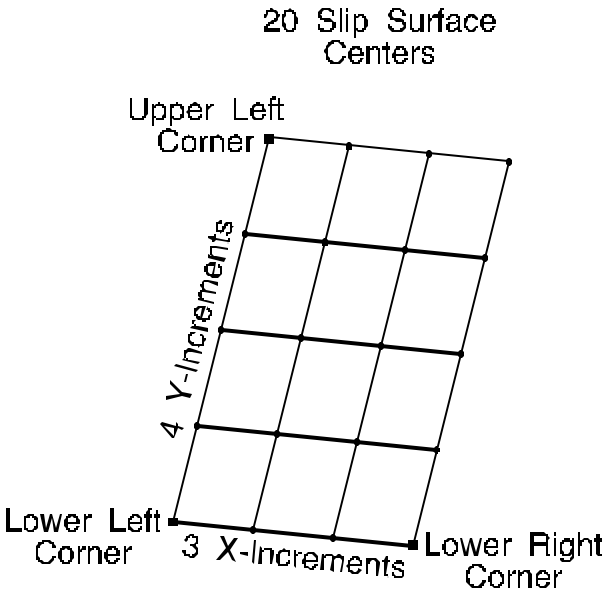
Comments

This command can only be chosen if you have selected the Grid & Radius slip surface option with the KeyIn Analysis Control command.

Points must be defined before they are specified as grid or radius corner points. However, Draw Slip Surface: Grid and Draw Slip Surface: Radius allow you to either snap to points or create points as you draw.

Figure 4.15 shows the position of the three corner points on the grid and the meaning of the increment values.

Figure 4.15 Definition of the Grid of Slip Surface Centers



The position and shape of the grid can be changed by moving the grid corner points using the Modify Objects command.

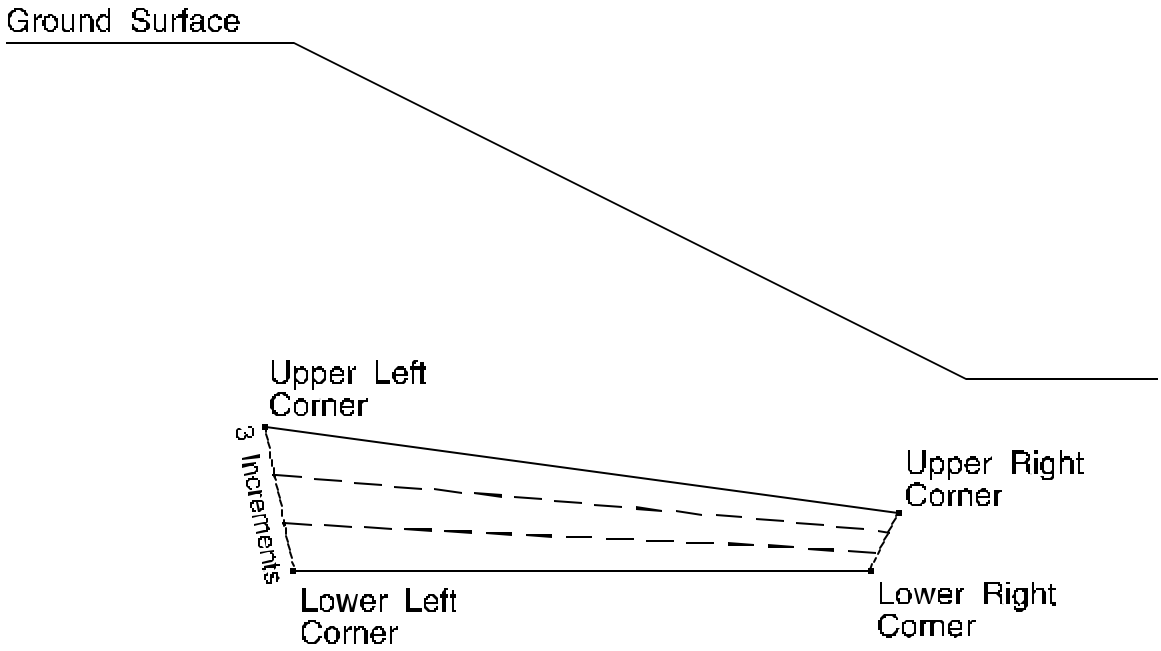
To define a single slip surface center, specify all three grid corner points as the same point number.

To define a series of centers along a non-vertical straight line, specify the upper left and lower left corner points as one point and the lower right corner point as another point. Enter the number of *x*-increments along the line, and set the number of *y*-increments to zero.

To define a series of centers along a non-horizontal straight line, specify the upper left corner as one point and the lower left and lower right corner points as another point. Enter the number of *y*-increments along the line, and set the number of *x*-increments to zero.

Figure 4.16 shows the position of the four radius corner points and the meaning of the increment values.

Figure 4.16 Definition of Four Radius Tangent Lines



The position and shape of the radius lines can be changed by moving the radius corner points using the Modify Objects command.

The radius lines do not have to be parallel to each other.

- **To define a single radius line for each slip surface center:**

  1. Specify the upper and lower left points as the same point number and specify the upper and lower right points as the same point number.

  2. Set the number of radius increments to zero.

- **To force all slip surfaces to pass through a single point:**

  1. Specify all four radius corner points as the same point number.

  2. Set the number of radius increments to zero.

- **To force all slip surfaces to pass through a series of points:**

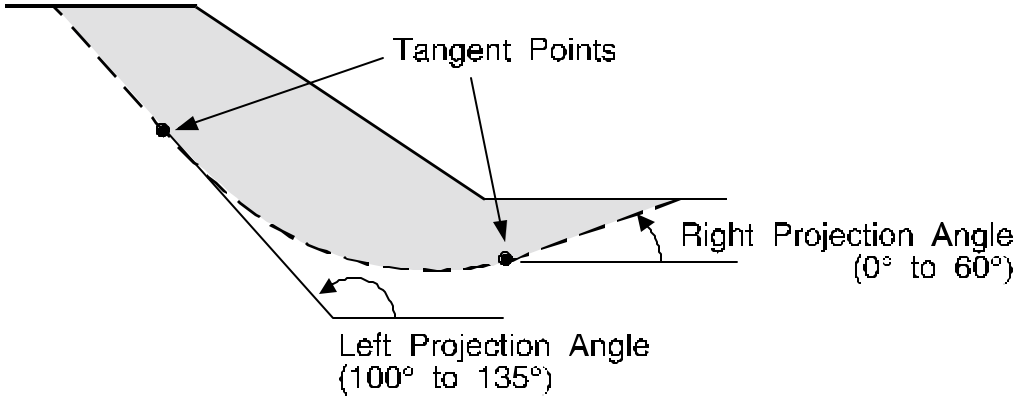
  1. Specify the lower left and right points as the same point number and specify the upper left and right points as the same point number.

  2. Set the number of radius increments to a value greater than zero.

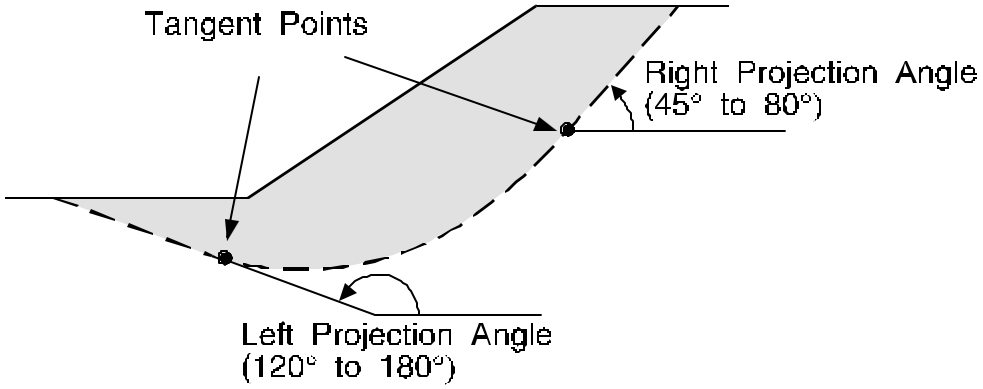
     Figure 4.17 illustrates the definition of the slip surface projection angles.

Figure 4.17 Definition of the Slip Surface Projection Angles

(a) Left to Right Problem



(b) Right to Left Problem



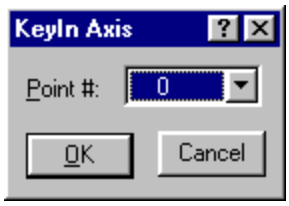
KeyIn Slip Surface Axis

☐ Selects the point about which to compute moment equilibrium.

The axis point is the point about which moment forces are summed to compute the moment equilibrium factor of safety. The Theory section explains the relationship between the point used to define the circular portion of the slip surface and the moment equilibrium axis.

An axis point must be defined if the Fully Specified or Block Specified slip surface option has been selected using the KeyIn Analysis Control command. The axis point is optional if the Grid & Radius option has been selected.

When you choose KeyIn Slip Surface Axis, the following dialog box appears:



➢ **To change the axis point:**

1. Click on the down-arrow to the right of the Point # edit box.

2. Select the desired point number.

3. Select OK.

Comments

The primary method of defining an axis point is by drawing it on the screen with Draw Slip Surface: Axis.

The axis point can be deleted by selecting 0 in the Point # drop-down list box or by choosing Modify Objects and deleting the axis point.

Methods that satisfy both moment and force equilibrium (e.g., Morgenstern-Price and GLE) are insensitive to the axis point used to sum moments. Methods that satisfy only moment or only force equilibrium can be slightly affected by the moment equilibrium point (see the Moment Axis section of Chapter 8 for more information). As a general rule, the axis point should be located approximately at the center of rotation of the slip surfaces.

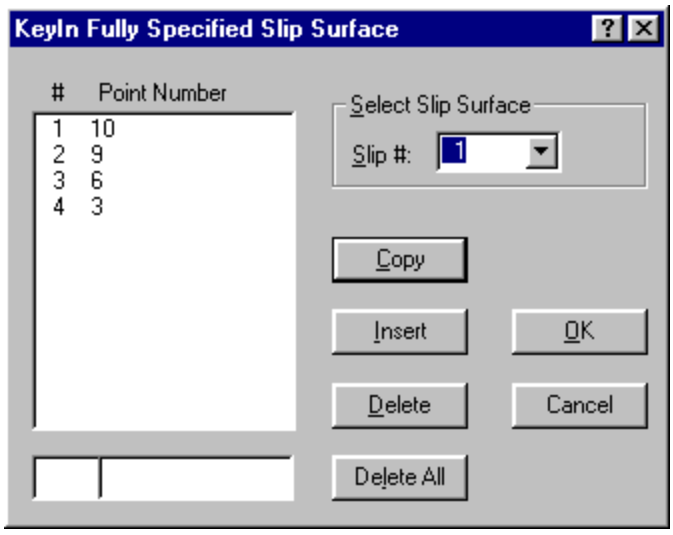
KeyIn Slip Surface Specified

☐ Defines the slip surfaces as piece-wise linear line segments.

Fully specified slip surfaces are slip surfaces made up of a series of line segments. Each slip surface must be specified individually by defining the points that make up the slip surface line.

This command can only be chosen if you have selected the Fully Specified slip surface option using the KeyIn Analysis Control command.

When you choose KeyIn Slip Surface Specified, the following dialog box appears:



The primary method of defining fully specified slip surfaces is by drawing them on the screen with the Draw Slip Surface: Specified command.

The main purpose of KeyIn Slip Surface Specified is to:

- View the point numbers contained in the slip surface lines.
- Define the sequences of points after the slip surfaces have been drawn.
- Delete fully specified slip surfaces.

- **To define or modify fully specified slip surfaces in the dialog box:**

  1. In the Select Slip Surface group box, select or type the slip surface number. If the slip surface already has been defined, the list box will be filled with the points contained in the slip surface.

  2. In the # edit box, type a number indicating where the point will be added on the slip surface (i.e., 1 will add the point to the beginning of the slip surface).

  3. In the Point Number edit box, type the point number to add to the slip surface.

  4. Select Copy to transfer the point data to the list box.

  5. Repeat Steps 2 to 4 for all points in the slip surface to define.

  6. To modify a point in the slip surface, click on the point in the list box with the left mouse button.

     The position of the point in the slip surface and the point number are copied into the edit boxes.

  7. Delete the point by selecting Delete or modify it by typing in a new point number and selecting Copy.

  8. Repeat Steps 1 to 7 for all slip surfaces to define.

  9. Select OK.

- **To insert a new point in the slip surface line:**

  1. Type a number in the # edit box to indicate where the point will be inserted in the slip surface.

  2. In the Point Number edit box, type the point number to insert in the slip surface.

  3. Select Insert.

     The new point is inserted in the list box. For example, if you insert a new point at the third position in a slip surface line, the point is inserted between the second and third points in the line.

  Points can also be inserted graphically by using the Draw Slip Surface: Specified command and clicking on the line where you want the point to be inserted.

  To delete all points in a slip surface line, select Delete All. To select multiple points in the list box for deletion, either press the CTRL key and click on each point to delete or press the SHIFT key and click on the first and last point to delete; then press the Delete button to delete the selected points.

  Points can be deleted graphically by choosing Modify Objects and deleting the points; the deleted points are then removed from the slip surface line data.

Comments

An axis point must be defined using Draw Slip Surface: Axis before any fully specified slip surfaces can be defined. Figure 4.18 shows the definition of an axis point and three fully specified slip surfaces.

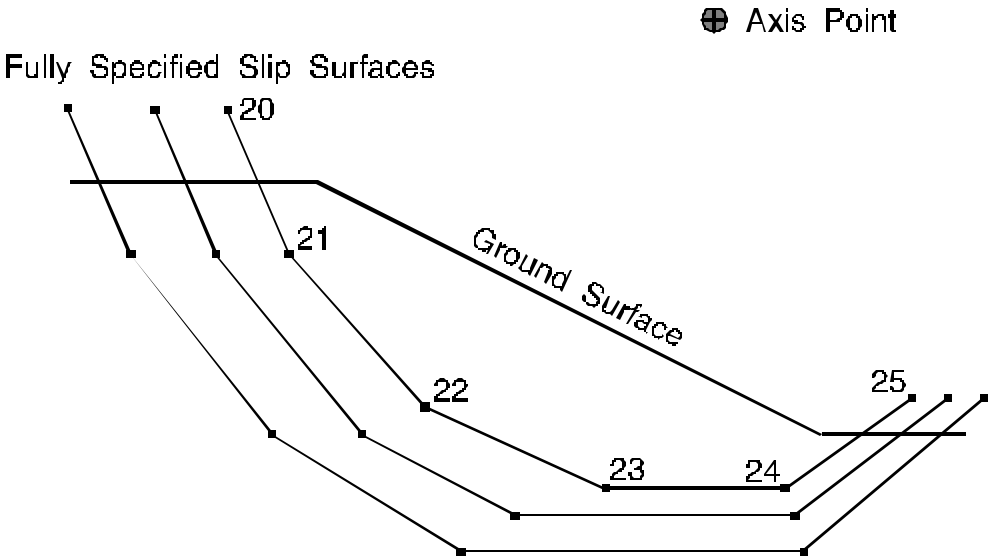
The position and shape of the fully specified slip surface can be modified by moving the slip surface points with the Modify Objects command.

The first and last endpoint of each fully specified slip surface must lie above the top of the geometry (i.e., Soil Line 1). If either endpoint lies underneath Soil Line 1, an error will be displayed when you choose the Tools Verify command.

Fully specified slip surfaces must be defined from left to right. In other words, the *x*-coordinate of each point in a slip surface must be greater than the *x*-coordinate of the previous point in the slip surface. Vertical line segments in a specified slip surface are not permitted.

The left and right endpoints of the fully specified slip surfaces must not extend beyond the boundaries of the geometry lines.

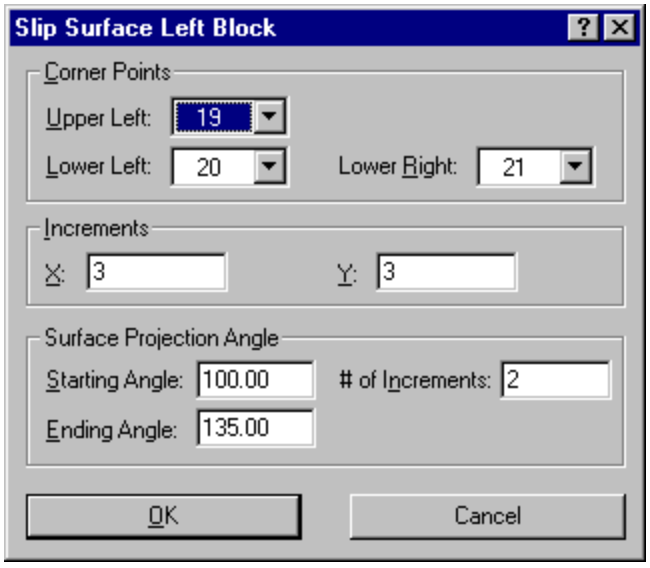
Figure 4.18 Definition of Fully Specified Slip Surfaces



KeyIn Slip Surface Left Block

☐ Defines the left block of intersection points for a generated piece-wise linear slip surface.

When you choose KeyIn Slip Surface Left Block, the following dialog box appears:



The left block points are used as the left intersection points of a generated linear slip surface. A block-specified slip surface consists of several line segments defined by two grids of intersection points. Slip surfaces are created by

connecting each point in the left block with each point in the right block, and then projecting each point to the surface at a series of specified projection angles.

The surface projection angles are defined by entering a range of angles and the number of increments used to subdivide the range.

The primary method of defining the left block is by drawing it on the screen with the Draw Slip Surface: Left Block command.

The main purpose of KeyIn Slip Surface Left Block is to:

- Specify an exact range of surface projection angles.
- View the points and increments numerically.
- Edit the point numbers and increments.

➢ **To modify the left block and left projection angle settings in the dialog box:**

1. To change any of the points in the Corner Points group box, click on the down-arrow to the right of one of the corner point edit boxes, and select the desired point number.
2. Repeat Step 1 for all block corner points to modify. Make sure that the points selected represent upper-left, lower-left, and lower-right corner points respectively.
3. Type the *x* and *y* block increments in the X and Y edit boxes.
4. Enter the left surface projection angle settings in the Surface Projection Angles group box.

   Starting Angle Specifies the starting, or minimum, surface projection angle.

   Ending Angle Specifies the ending, or maximum, surface projection angle.

   Number of Increments Specifies the number of increments between the Starting Angle and Ending Angle. The number of projection angles is one greater than the Number of Increments. For example, if 3 increments are specified and the Starting and Ending Angles are 100 and 130 degrees respectively, the resulting projection angles are 100, 110, 120, and 130 degrees. If 0 increments are specified, than the Ending Angle is ignored and only the Starting Angle is used.

   If the direction of slip surface movement is from left to right, the range of projection angles must be between 100º and 135º; if the direction is from right to left, the range of projection angles must be between 120º and 180º.

5. Select OK.

➢ **To define a single block point:**

- Specify all three block corner points as the same point number.

➢ **To define a series of block points along a non-vertical straight line:**

1. Specify the upper left and lower left corner points as one point and the lower right corner point as another point.
2. Enter the number of *x*-increments along the line.
3. the number of *y*-increments to zero.

➢ **To define a series of block points along a non-horizontal straight line:**

1. Specify the upper left corner as one point and the lower left and lower right corner points as another point.
2. Enter the number of *y*-increments along the line.
3. Set the number of *x*-increments to zero.

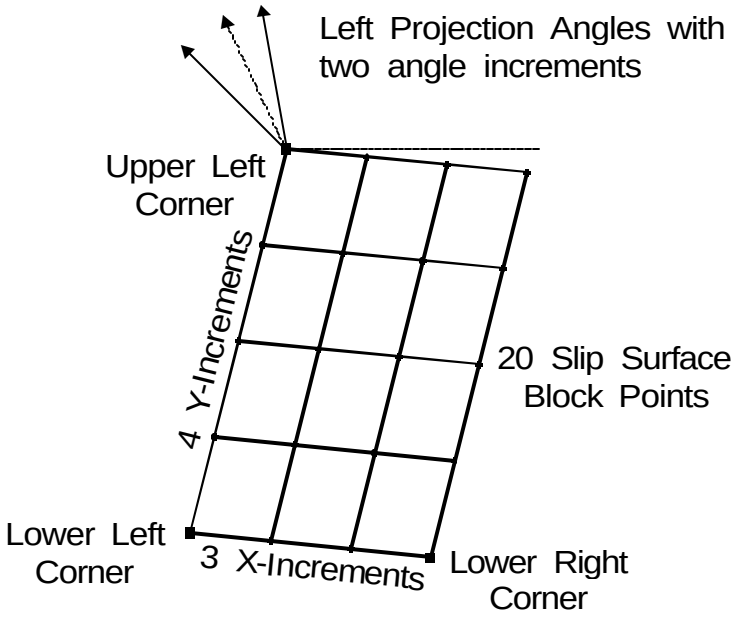
Comments

This command can only be chosen if you have selected the Block Specified slip surface option using the KeyIn Analysis Control command.

Points must be defined before they are specified as block corner points. However, Draw Slip Surface: Left Block allows you to either snap to points or create points as you draw.

Figure 4.19 shows the position of the three block corner points, the projection angles, and the meaning of the increment values.

Figure 4.19 Definition of the Left Slip Surface Block



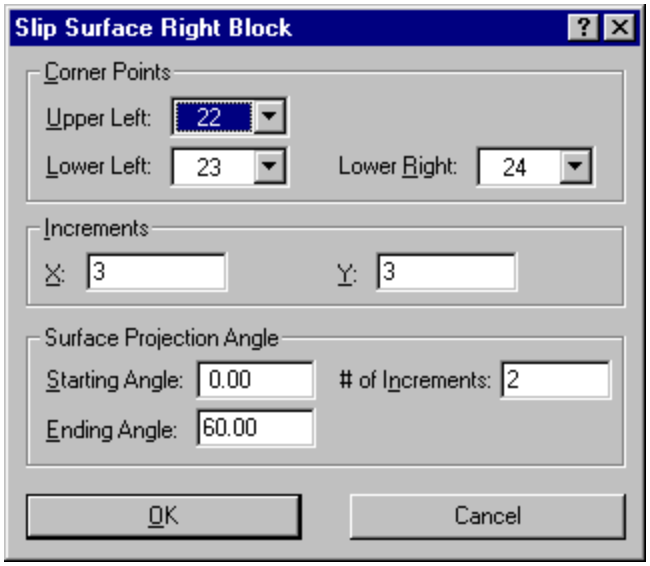
The position and shape of the block can be modified by moving the block corner points with the Modify Objects command.

Choose the KeyIn Slip Surface: Right Block or Draw Slip Surface: Right Block command to specify the right slip surface block of intersection points.

KeyIn Slip Surface Right Block

☐ Defines the right block of intersection points for a generated piece-wise linear slip surface.

When you choose KeyIn Slip Surface Right Block, the following dialog box appears:



The right block points are used as the right intersection points of a generated linear slip surface. A block-specified slip surface consists of several line segments defined by two grids of intersection points. Slip surfaces are created by connecting each point in the left block with each point in the right block, and then projecting each point to the surface at a series of specified projection angles.

The surface projection angles are defined by entering a range of angles and the number of increments used to subdivide the range.

The primary method of defining the right block is by drawing it on the screen with the Draw Slip Surface: Right Block command.

The main purpose of KeyIn Slip Surface Right Block is to:

- Specify an exact surface projection angle.
- View the points and increments numerically.
- Edit the point numbers and increments.

➢ **To modify the right block and right projection angle in the dialog box:**

1. To change any of the points in the Corner Points group box, click on the down-arrow to the right of one of the corner point edit boxes, and select the desired point number.

2. Repeat Step 1 for all block corner points to modify. Make sure that the points selected represent upper-left, lower-left, and lower-right corner points respectively.

3. Type the *x* and *y* block increments in the X and Y edit boxes.

4. Enter the right surface projection angle settings in the Surface Projection Angles group box.

   Starting Angle Specifies the starting, or minimum, surface projection angle.

Ending Angle Specifies the ending, or maximum, surface projection angle.

Number of Increments Specifies the number of increments between the Starting Angle and Ending Angle. The number of projection angles is one greater than the Number of Increments. For example, if 3 increments are specified and the Starting and Ending Angles are 0 and 60 degrees respectively, the resulting projection angles are 0, 20, 40, and 60 degrees. If 0 increments are specified, than the Ending Angle is ignored and only the Starting Angle is used.

If the direction of slip surface movement is from left to right, the range of projection angles must be between 0º and 60º; if the direction is from right to left, the range of projection angles must be between 45º and 80º.

5. Select OK.

➢ **To define a single block point:**

- Specify all three block corner points as the same point number.

➢ **To define a series of block points along a non-vertical straight line:**

1. Specify the upper left and lower left corner points as one point and the lower right corner point as another point.
2. Enter the number of *x*-increments along the line.
3. Set the number of *y*-increments to zero.

➢ **To define a series of block points along a non-horizontal straight line:**

1. Specify the upper left corner as one point and the lower left and lower right corner points as another point.
2. Enter the number of *y*-increments along the line:
3. Set the number of *x*-increments to zero.

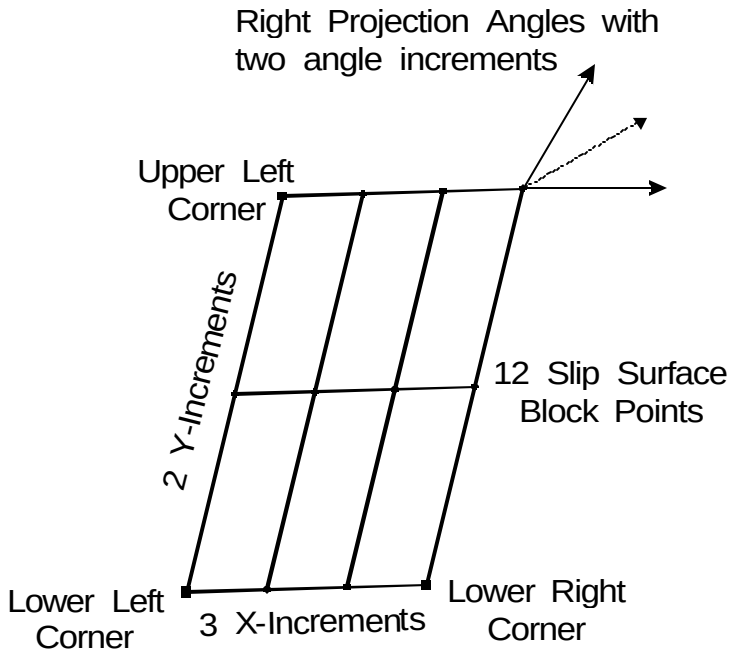
Comments

This command can only be chosen if you have selected the Block Specified slip surface option with the KeyIn Analysis Control command.

Points must be defined before they are specified as block corner points. However, Draw Slip Surface: Right Block allows you to either snap to points or create points as you draw.

Figure 4.20 shows the position of the three block corner points, the projection angles, and the meaning of the increment values.

Figure 4.20 Definition of the Right Slip Surface Block



The position and shape of the block can be modified by moving the block corner points with the Modify Objects command.

Choose the KeyIn Slip Surface: Left Block or Draw Slip Surface: Left Block command to specify the left slip surface block of intersection points.

KeyIn Slip Surface Limits

☐ Defines the limits within which the slip surface must intersect the top soil layer.

Each trial slip surface must intersect the top soil layer between the left and right limits. SOLVE will not analyze any slip surface that intersects the top soil line beyond these limits. The KeyIn Slip Surface Limits command lets you specify these limits.

When you choose KeyIn Slip Surface Limits, the following dialog box appears:

➢ **To specify the slip surface limits:**

1. Select Line 1 Extents if you wish to use the extents of the top soil layer as the slip surface limits. By default, SLOPE/W uses the Line 1 Extents option.

2. If you wish to specify your own *x*-coordinates as the slip surface limits, select the Specified X-Coordinates option. Type the minimum *x*-coordinate in the Min. X edit box, and type the maximum *x*-coordinate in the Max. X edit box.

   SOLVE will ignore all slip surfaces that intersect the top soil outside of these *x*-coordinates.

3. Select OK.

Comments

The minimum slip surface limit is displayed above Soil Line 1 as the symbol, . The maximum slip surface limit is displayed above Soil Line 1 as the symbol, .

You can also use the Draw Slip Surface: Limits command to define the slip surface limits by dragging the symbols along the top soil line.

KeyIn Pore Pressure: Water Pressure

☐ Defines the pore-water pressure conditions.

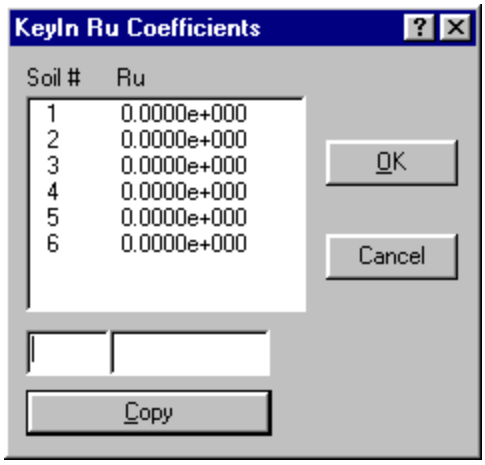
KeyIn Pore-Water Pressure defines the pore-water pressure according to the method selected with the KeyIn Analysis Control command. Depending on which method has been selected, SLOPE/W obtains the pore-water pressure data for one of the following methods:

(none)

The KeyIn Pore-Water Pressure command is disabled if no pore-water pressure is specified for the problem.

Ru Coefficients

The following dialog box appears when you select KeyIn Pore-Water Pressure:



The default r_u values for each defined soil are displayed in the list box.

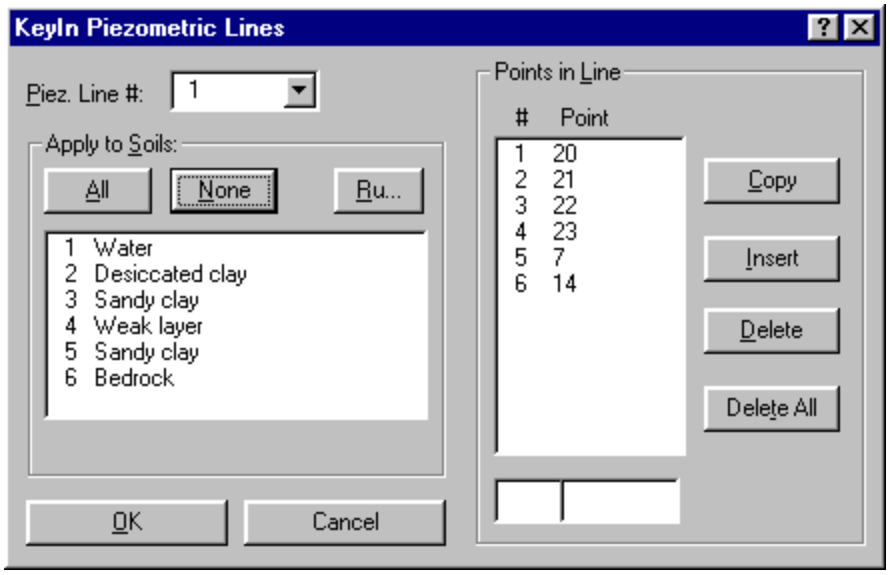
➢ **To define new r_u values for each soil:**

1. In the list box, select the soil number and r_u value to change.
2. In the Ru edit box, type the new r_u value for the soil number.
3. Select Copy to transfer the point data to the list box.
4. Repeat Steps 1 to 3 for all r_u values that you wish to change.
5. Select OK.

See the KeyIn Analysis Control command in this chapter for further discussion on r_u coefficients.

Piezometric Lines / Ru

The following dialog box appears when you select KeyIn Pore-Water Pressure:



The primary method of defining piezometric lines is by drawing them on the screen with Draw Pore-Water Pressure.

The main purpose of KeyIn Pore-Water Pressure is to:

- Check which soil layers are applied to individual piezometric lines.
- Modify the piezometric line data.
- Delete piezometric lines.
- Apply R_u values in addition to piezometric lines.

➢ **To define or modify piezometric line data in the dialog box:**

1. In the Piez. Line # edit box, type the piezometric line number to define. A list of piezometric lines already defined can be obtained by clicking the arrow to the right of the edit box. Select one of these numbers if you wish to modify a piezometric line that has already been defined.

   If the piezometric line has already been defined, the soils applied to the piezometric line will be selected in the Apply To Soils list box, and the Points In Line list box will be filled with the points contained in the line.

2. In the Apply To Soils list box, select the soils to apply to the piezometric line by clicking on each soil in the list box. Click on the soil again to unselect it. Select All to apply all soils to the piezometric line, or select None to unselect all soils in the list box.

3. In the Points In Line group box, type a number in the # edit box indicating where the point will be added on the line (i.e., 1 will add the point to the beginning of the line).

4. In the Point Number edit box, type the point number to add to the line.

5. Select Copy to transfer the point data to the list box.

6. Repeat Steps 3 to 5 for all points in the line.

7. To modify a point in the piezometric line, click on the point in the list box with the left mouse button.

   The position of the point in the line and the point number are copied into the edit boxes.

8. Delete the point by selecting Delete or modify it by typing in a new point number and selecting Copy.

9. Repeat Steps 1 to 8 for all piezometric lines to define.

10. Select OK.

➢ **To insert a new point in the piezometric line:**

1. Type a number in the # edit box to indicate where the point will be inserted in the line.

2. In the Point Number edit box, type the point number to insert in the line.

3. Select Insert.

   The new point is inserted in the list box. For example, if you insert a new point at the third position in a line, the point is inserted between the second and third points in the line.

Points can also be inserted graphically by using the Draw Pore-Water Pressure command and clicking at the position on the piezometric line where you want the point to be inserted.

➢ **To delete all points in a piezometric line:**

- Select Delete All.

The piezometric line will be deleted when you select OK if there are no points in the line or if no soils are applied to the line.

➢ **To select multiple points in the list box for deletion:**

- Press the CTRL key and click on each point to delete.

  -- or --

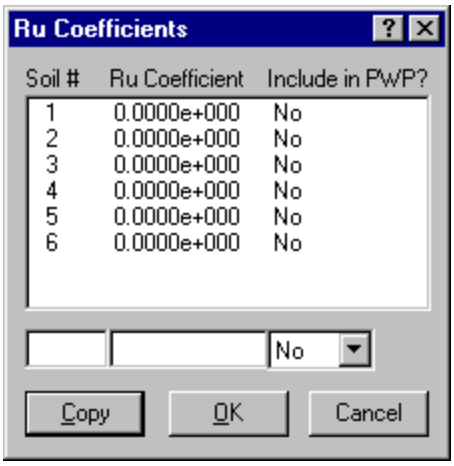
- Press the SHIFT key and click on the first and last point to delete, then press the Delete button to delete the selected points.

To delete or move points graphically, choose the Modify Objects command. Deleting points will remove them from the piezometric line. Moving points will also move the piezometric lines attached to the points.

R_u coefficients can be included in the pore-water pressure calculations along with piezometric lines.

➢ **To include the R_u coefficients with the pore-water pressure calculations:**

1. Select Ru in the Piezometric Lines dialog box. The following dialog box appears:



2. Select the desired soil number by clicking on the soil in the list box.

   The soil number and its R_u coefficient are copied into the edit boxes. If the soil is to be included in the P.W.P. calculations, Yes is selected in the drop-down list box; otherwise, No is selected.

3. Enter the R_u coefficient value in the R_u Coefficient edit box.

4. Include the soil R_u coefficient in the P.W.P. calculations by selecting Yes in the drop-down list box. To remove the R_u coefficient from the calculations, select No.

5. Select Copy to transfer the data to the list box.

6. Repeat Steps 2 to 5 for all desired soils.

7. Select OK.

   An asterix appears in the Piezometric Lines dialog box beside all soils that have R_u coefficients included in the P.W.P. calculations.

Each piezometric line must start at the left extremity of the problem and extend across the problem to the right extremity. In other words, the starting and ending *x*-coordinates of each piezometric line must match the left and right boundaries of the problem. The Tools Verify command enforces this rule by modifying the *x*-coordinates of the line endpoints if necessary.

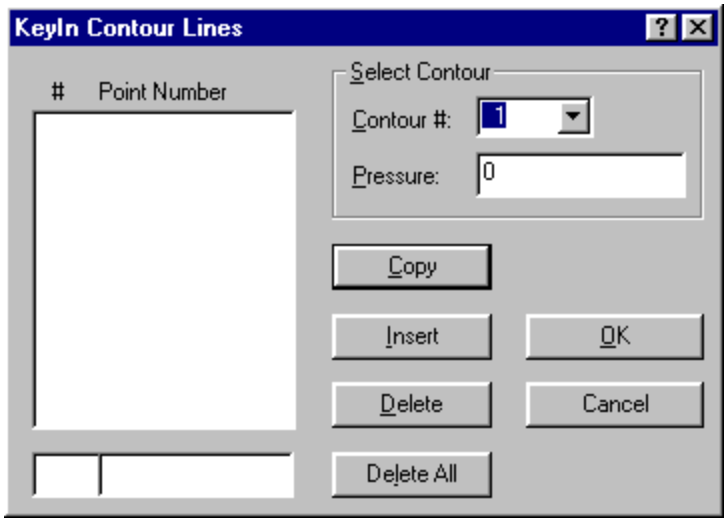
Piezometric lines must be defined from left to right. In other words, the *x*-coordinate of each point in a line must be greater than the *x*-coordinate of the previous point in the line. Vertical segments in a line are not permitted.

Piezometric lines are displayed on the drawing as blue dashed lines.

See the KeyIn Analysis Control command in this chapter for further discussion on piezometric lines.

Contours

The following dialog box appears when you select KeyIn Pore-Water Pressure:



➢ **To define or modify contour line data in the dialog box:**

1. In the Select Contour group box, select or type the contour number in the Contour # edit box. If the contour number already has been defined, the list box will be filled with the points contained in the contour.
2. In the Pressure edit box, type the pore-water pressure along the contour line.
3. In the # edit box, type a number indicating where the point will be added on the contour (i.e., 1 will add the point to the beginning of the contour).
4. In the Point Number edit box, type the point number to add to the contour line.
5. Select Copy to transfer the point data to the list box.
6. Repeat Steps 3 to 5 for all points in the contour line.
7. To modify a point in the contour line, click on the point in the list box with the left mouse button.

   The position of the point in the contour and the point number are copied into the edit boxes.
8. Delete the point by selecting Delete or modify it by typing in a new point number and selecting Copy.
9. Repeat Steps 1 to 8 for all contour lines to define.
10. Select OK.

➢ **To delete all points in a contour:**

- Select Delete All.

➢ **To insert a new point in the contour line:**

1. Type a number in the # edit box to indicate where the point will be inserted in the line.
2. In the Point Number edit box, type the point number to insert in the line.
3. Select Insert.

   The new point is inserted in the list box. For example, if you insert a new point at the third position in a line, the point is inserted between the second and third points in the line.

Points can also be inserted graphically by using the Draw Pore-Water Pressure command and clicking at the position on the contour line where you want the point to be inserted.

➢ **To delete all points in a contour line:**

- Select Delete All.

➢ **To select multiple points in the list box for deletion:**

- Press the CTRL key and click on each point to delete.

  -- or --

- Press the SHIFT key and click on the first and last point to delete; then press the Delete button to delete the selected points.

To delete or move points graphically, choose the Modify Objects command. Deleting points will remove them from the contour line. Moving points will also move the contour lines attached to the points.

Each contour line must start at the left extremity of the problem and extend across the problem to the right extremity. In other words, the starting and ending *x*-coordinates of each contour line must match the left and right boundaries of the problem. The Tools Verify command enforces this rule by modifying the *x*-coordinates of the line endpoints if necessary.

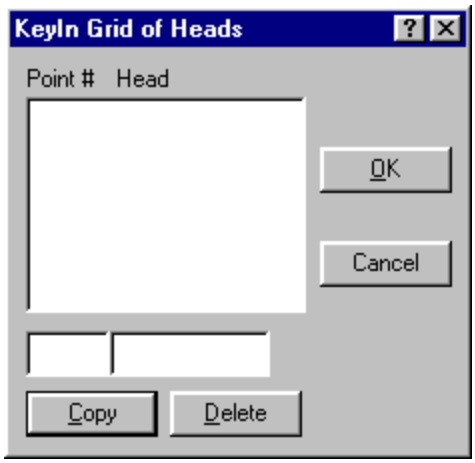
Contour lines must be defined from left to right. In other words, the *x*-coordinate of each point in a line must be greater than the *x*-coordinate of the previous point in the line. Vertical segments in a line are not permitted.

Contour lines are displayed on the drawing as blue dashed lines.

See the KeyIn Analysis Control command in this chapter for further discussion on pore-water pressure contours.

Grid of Heads, Grid of Pressures, and Grid of Ru Coefficients

The following dialog box appears when you select KeyIn Pore-Water Pressure and the Grid of Heads P.W.P. option has been selected (similar dialog boxes appear for the Grid of Pressures and Grid of R_u Coefficients options):



➢ **To define or modify a grid of heads, pressures, or r_u coefficients:**

1. In the Point # edit box, type the point number at which to define the head, pressure, or r_u value.
2. In the Head, Pressure, or R_u edit box, type the appropriate value.
3. Select Copy to transfer the data to the list box.
4. To modify a point in the grid, click on the point in the list box with the left mouse button.

   The point number and the grid value are copied into the edit boxes.
5. Delete the point from the grid by selecting Delete or modify it by typing in a new head, pressure, or r_u value and selecting Copy.

   To select multiple points in the list box for deletion, either press the CTRL key and click on each point to delete or press the SHIFT key and click on the first and last point to delete; then press the Delete button to delete the selected points.
6. Repeat Steps 1 to 5 for all points at which to specify head, pressure, or r_u values.
7. Select OK.

Comments

Grid points are displayed on the drawing as blue triangles.

Pressure head must be defined in the units of length used in the problem, such as metres or feet.

See the KeyIn Analysis Control command in this chapter for further discussion on the grid of heads, grid of pressures, and grid of r_u coefficients.

SEEP/W Heads and SIGMA/W Pressures

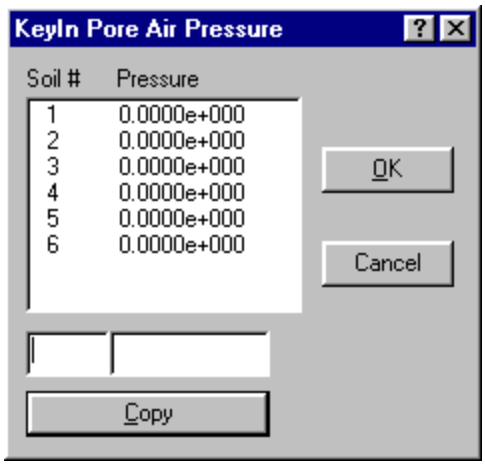
The KeyIn Pore-Water Pressure command is disabled if the selected P.W.P. option is SEEP/W Heads or SIGMA/W Pressures, since the pore-water pressure data is obtained directly from the finite element analysis. See the KeyIn

Analysis Control command in this chapter for further discussion on using finite element-computed pore-water pressures.

KeyIn Pore Pressure: Air Pressure

☐ Specifies pore-air pressure for each soil layer.

When you choose KeyIn Air Pressure, the following dialog box appears:



The pore-air pressure is defined for each soil type. It is applicable in such problems as the placement of earth fills, where the pore-air pressure may be greater than atmosphere for a time during construction.

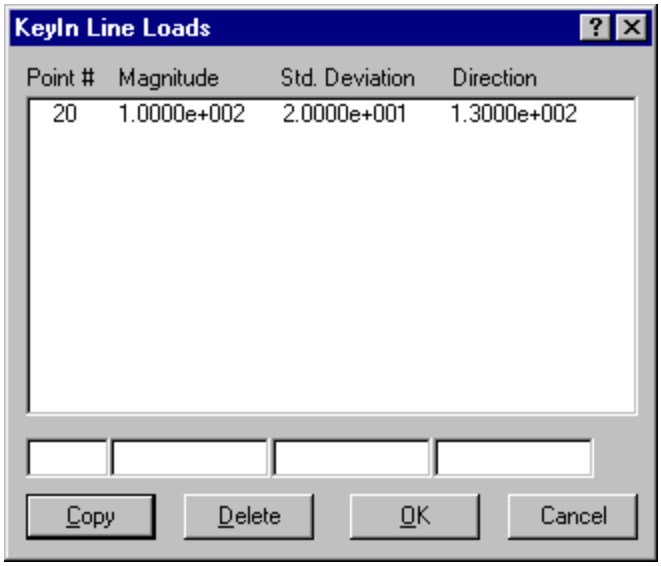
➢ **To define pore-air pressure:**

1. Select the line in the list box containing the soil number for which to define pore-air pressure. The information is copied into the edit boxes.
2. In the Pressure edit box, type the pore-air pressure for the soil.
3. Select Copy to transfer the pore-air pressure data to the list box.
4. Repeat Steps 1 to 3 for all soils for which to define pore-air pressure.
5. Select OK.

KeyIn Load: Line Loads

☐ Sets the position, magnitude, and direction of concentrated loads.

When you choose KeyIn Line Loads, the following dialog box appears:



Concentrated line loads can exist at points. The loads are defined by the location of the point, the magnitude, and the direction of the load.

Line loads can be used to simulate any concentrated load, such as a structural load or the resultant earth pressure on a retaining wall.

The primary method of defining line loads is by drawing them on the screen with Draw Line Loads. The main purpose of KeyIn Line Loads is to edit or delete line loads after they have been drawn.

➢ **To define or modify a line load:**

1. In the Point # edit box, type the point number at which to define the line load.
2. Type the magnitude of the line load in the Magnitude edit box.
3. Type the direction (in degrees) of the line load in the Direction edit box.
4. Select Copy to transfer the data to the list box.
5. To modify a line load, click on the point in the list box with the left mouse button.

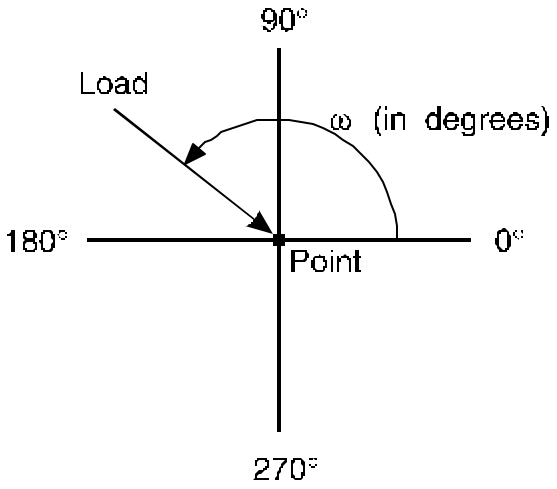
   The point number, magnitude, and direction of the line load are copied into the edit boxes.
6. Delete the line load by selecting Delete or modify it by typing in a magnitude or direction and selecting Copy.

   To select multiple line loads in the list box for deletion, either press the CTRL key and click on each line load to delete or press the SHIFT key and click on the first and last line load to delete; then press the Delete button to delete the selected line loads.
7. Repeat Steps 1 to 6 for all line loads to define or modify.
8. Select OK.

To delete or move line loads graphically, choose the Modify Objects command. Deleting the point defining the line load will remove the line load. Moving the line load point will also move the line load. To graphically change the line load direction, choose Draw Line Loads.

Figure 4.21 shows the sign convention for line loads.

Figure 4.21 Definition of Line Loads



Comments

Line loads affect the forces on the slice to which the load is applied. Both the normal at the base of the slice and forces between adjacent slices are affected.

When you are evaluating the detailed forces on a slice, be aware that more than one line load may affect a particular slice.

Line loads must be applied within the sliding mass to be included in the stability calculations. Line loads outside the sliding mass are ignored.

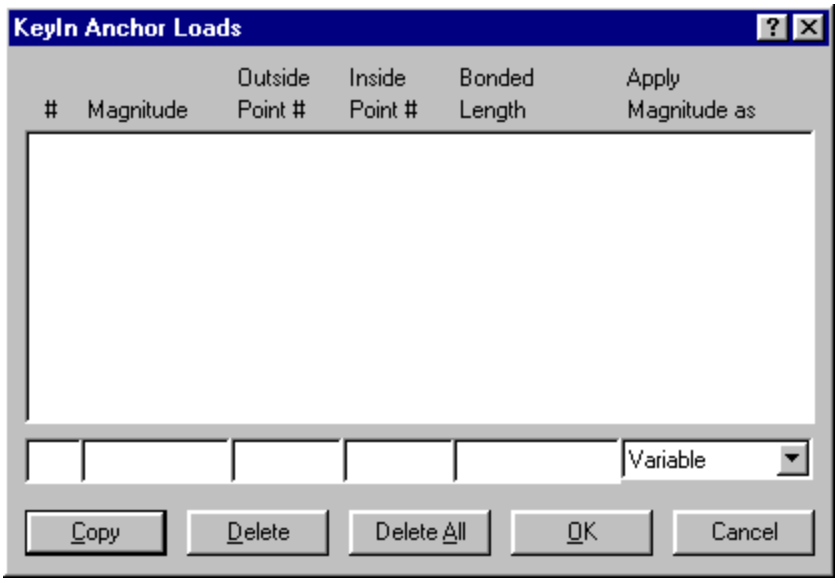
KeyIn Load: Anchor Loads

☐ Defines anchors acting as concentrated loads within the soil.

Anchor loads are similar to concentrated line loads (see KeyIn Load: Line Loads). Using anchor loads, however, allows you to make the load magnitude vary depending on where the slip surface intersects the anchor. Another difference is that anchor loads act at the slice base, whereas line loads act at the point where they are defined.

Anchor loads are useful for modelling features such as ground anchors, soil nails, or geofabric reinforcement.

When you choose KeyIn Anchor Loads, the following dialog box appears:



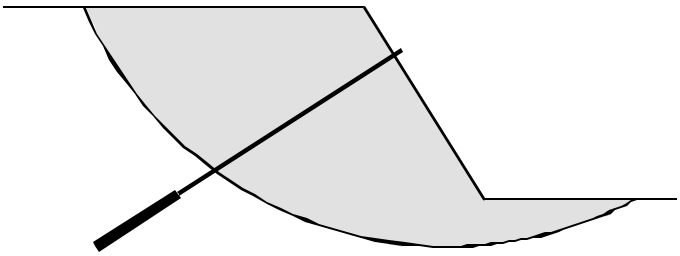
The primary method of defining anchor loads is by drawing them on the screen with Draw Anchor Loads. The main purpose of KeyIn Anchor loads is to edit or delete anchor loads after they have been drawn.

Anchor loads are defined by specifying two points that form a line segment. The first point is defined above the geometry at the position where the anchor is inserted. The second point is defined inside the soil strata at the end of the anchor. A portion of the anchor line is designated as the *bonded length*, representing the part of the anchor that is bonded to the soil.

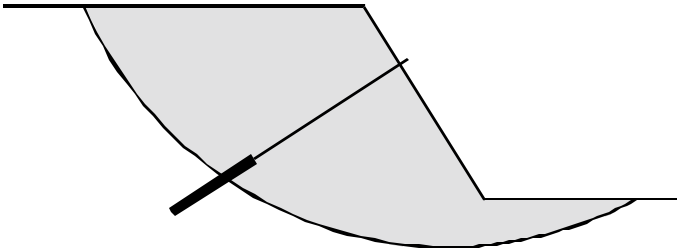
SLOPE/W applies the entire specified anchor load when a slip surface intersects the unbonded portion of the anchor. If the anchor lies completely within the sliding mass, it is ignored. When the slip surface intersects the bonded portion of the anchor, the anchor is ignored if the anchor load is being applied as a constant. If it is being applied variably, then a percentage of the specified magnitude is applied, as shown in Figures 4.22 and 4.23.

Figure 4.22 Definition of Anchor Loads when Applied with Constant Magnitude

(a) Full anchor load is applied



(b) Anchor load is ignored



(c) Anchor load is ignored

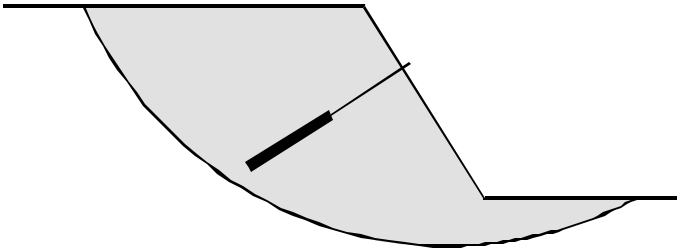
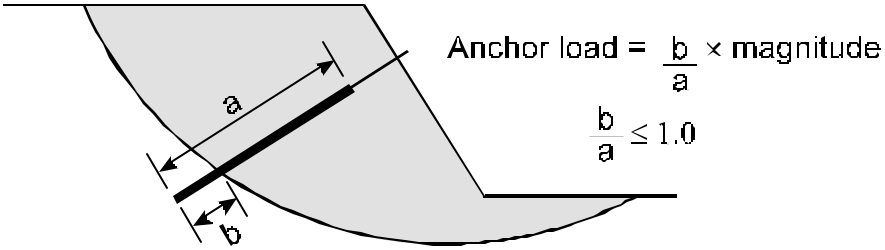


Figure 4.23 Definition of Anchor Loads when Applied with Variable Magnitude



➢ **To define or modify an anchor load:**

1. In the # edit box, type the anchor load number.
2. Type the magnitude of the anchor load in the Magnitude edit box.
3. In the Outside Point # edit box, type the point number outside the soil stratigraphy at which the anchor begins.
4. In the Inside Point # edit box, type the point number inside the soil stratigraphy at which the bonded portion of the anchor ends.
5. In the Bonded Length edit box, type the distance from the Inside Point to the position on the anchor where the bonding ends.
6. In the Apply Magnitude As drop-down list box, select Constant to ignore the bonded anchor portion or Variable to consider the bonded anchor portion when determining the intersection of the slip surface and the anchor.
7. Select Copy to transfer the data to the list box.
8. To modify an anchor load, click on the anchor in the list box with the left mouse button.

   The anchor number, magnitude, inside point, outside point, bonded length, and method of application are copied into the edit boxes.

9. Delete the anchor load by selecting Delete or modify it by typing in new values in the edit boxes and selecting Copy.

   To select multiple anchor loads in the list box for deletion, either press the CTRL key and click on each anchor load to delete or press the SHIFT key and click on the first and last anchor load to delete; then press the Delete button to delete the selected anchor loads.

10. Repeat Steps 1 to 9 for all anchor loads to define or modify.

11. Select OK.

To delete or move anchor loads graphically, choose the Modify Objects command. Deleting one of the points defining the anchor load will remove the anchor load. Moving an anchor load point will also move the anchor load. To graphically change the anchor load bonded length, choose Draw Anchor Loads.

Comments

Anchor loads affect the forces on the slice to which the load is applied. Both the normal at the base of the slice and forces between adjacent slices are affected.

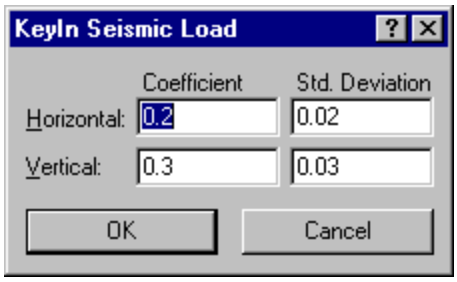
The anchor load is applied to the slice base that intersects the anchor line of action.

KeyIn Load: Seismic Load

☐ Sets horizontal and vertical coefficients representing a seismic force.

The KeyIn Seismic Load command can be used to simulate the effects of forces created by seismic or earthquake accelerations. SLOPE/W applies the seismic forces to the centroid of each slice equal to the slice weight multiplied by the seismic coefficients. For a complete discussion of applying seismic loads in SLOPE/W, see the Modelling Guidelines section.

When you choose KeyIn Seismic Load, the following dialog box appears:



➢ **To define a seismic load:**

1. To apply a horizontal seismic load, type the value of the seismic load in the Horizontal Coefficient edit box.

   If you have selected a Probabilistic Analysis with KeyIn Analysis Control, type a standard deviation for the Horizontal Coefficient in the Horizontal Std. Deviation edit box.

2. To apply a vertical seismic load, type the value of the seismic load in the Vertical Coefficient edit box.

   If you have selected a Probabilistic Analysis with KeyIn Analysis Control, type a standard deviation for the Vertical Coefficient in the Vertical Std. Deviation edit box.

3. Select OK.

Comments

The Vertical Coefficient can be specified as a zero, positive or negative value. A positive coefficient signifies that the vertical force is applied in the same direction as the weight of the slice, (i.e., downward), while a negative coefficient signifies the vertical force is applied in the opposite direction (i.e., upward).

The Horizontal Coefficient must be specified as a zero or positive value. A positive horizontal coefficient signifies that the additional force is acting horizontally in the same direction as the movement of the slope.

Materials that use the No Strength soil model (i.e., c=0 and $f = 0$) are not included in the slice weight when computing the seismic load. For example, if a slice is submerged under water, the weight of the water above the slice is not included in the slice weight used in the seismic load calculation.

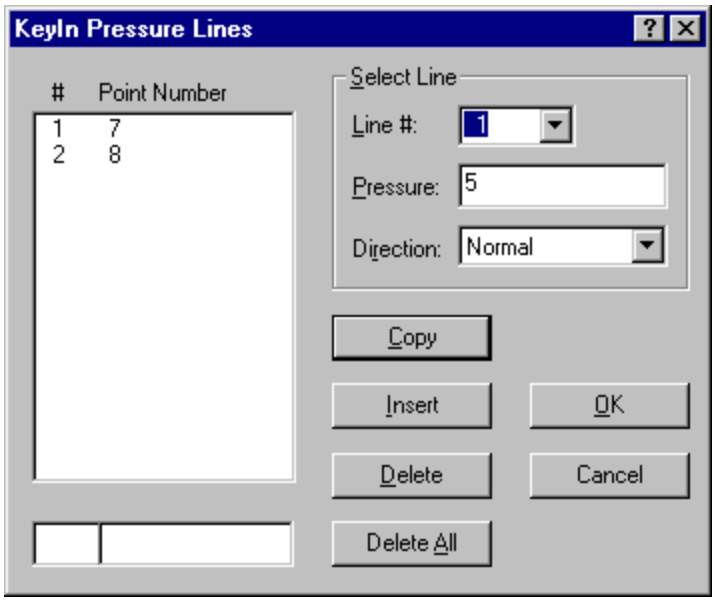
A free-body diagram can be displayed for each slice using the View Slice Forces command in CONTOUR. The horizontal seismic force vector is displayed as a horizontal force vector applied to the center of the slice. The vertical seismic force is integrated into the weight vector of the slice and is not displayed separately.

KeyIn Pressure Lines

☐ Defines pressure lines applied at the top soil surface.

Pressure lines are used to simulate a pressure applied over a portion of the soil surface (e.g., to model a footing on the ground surface). Unlike line loads, which are a concentrated force applied at one point, pressure lines are applied over a region. The magnitude of the applied pressure is computed by multiplying the specified pressure by the vertical distance between the pressure line and the soil surface. The direction of the applied pressure can be specified as normal to the ground surface or vertical.

When you choose KeyIn Pressure Lines, the following dialog box appears:



The primary method of defining pressure lines is by drawing them on the screen with Draw Pressure Lines.

The main purpose of KeyIn Pressure Lines is to:

- Check the data for individual pressure lines.
- Modify the pressure line data.

- Delete pressure lines.

➢ **To define or modify pressure line data in the dialog box:**

1. In the Select Line group box, select the pressure line number from the drop-down list box or type in a new pressure line number. If the specified pressure line already contains points, the list box will be filled with the points contained in the line.

2. In the Pressure edit box, type the amount of pressure to apply on the portion of the soil surface lying directly underneath the pressure line.

3. In the Direction drop-down list box, select Normal if you wish to apply the pressure at an angle normal to the soil surface. Select Vertical to always apply the pressure vertically down on the soil surface.

4. In the # edit box, type a number indicating where the point will be added on the pressure line (i.e., 1 will add the point to the beginning of the pressure line).

5. In the Point Number edit box, type the point number to add to the pressure line.

6. Select Copy to transfer the point data to the list box.

7. Repeat Steps 4 to 6 for all points in the pressure line to define.

8. To modify a point in the pressure line, click on the point in the list box with the left mouse button.

   The position of the point in the pressure line and the point number are copied into the edit boxes.

9. Delete the point by selecting Delete or modify it by typing in a new point number and selecting Copy.

10. Repeat Steps 1 to 9 for all pressure lines to define.

11. Select OK.

➢ **To insert a new point in the pressure line:**

1. Type a number in the # edit box to indicate where the point will be inserted in the line.

2. In the Point Number edit box, type the point number to insert in the line.

3. Select Insert.

   The new point is inserted in the list box. For example, if you insert a new point at the third position in a line, the point is inserted between the second and third points in the line.

Points can also be inserted graphically by using the Draw Pressure Lines command and clicking at the position on the line where you want the point to be inserted.

To delete all points in a line, select Delete All. To select multiple points in the list box for deletion, either press the CTRL key and click on each point to delete or press the SHIFT key and click on the first and last point to delete; then press the Delete button to delete the selected points.

To delete or move surface pressure line points graphically, choose the Modify Objects command. Deleting one of the points defining the surface pressure line will remove the pressure line. Moving a pressure line point will also move the pressure line.

Comments

The area underneath each pressure line and above Soil Line 1 is shaded with a cross-hatch pattern to indicate where the surface pressure will be applied on Soil Line 1.

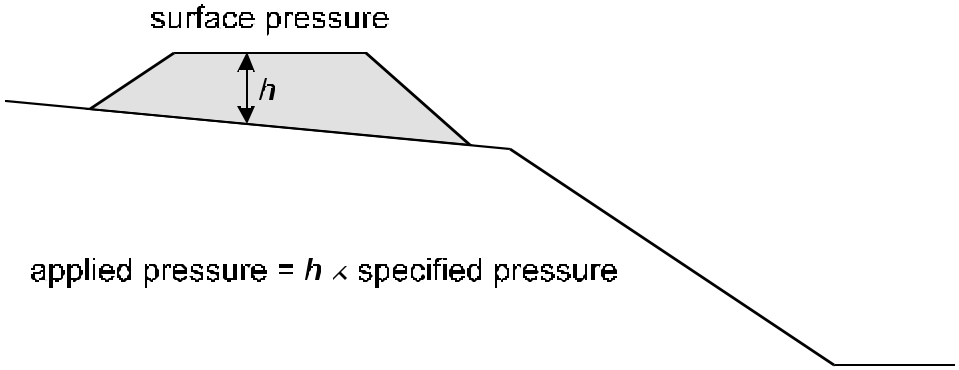
Pressure lines must be defined from left to right. In other words, the *x*-coordinate of each point in a line must be greater than the *x*-coordinate of the previous point in the line. Vertical segments in a line are not permitted.

Pressure lines must not cross or overlap with other pressure lines. All pressure lines must lie above the top soil line (Soil Line 1). Choose the Tools Verify command to help you verify that the surface pressure lines have been defined correctly.

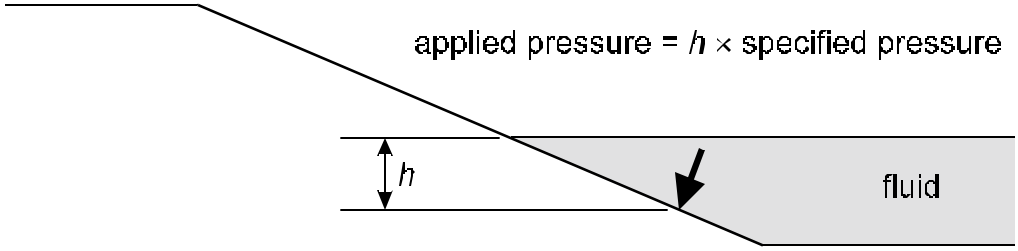
Figure 4.24 illustrates how surface pressures are applied in the normal and vertical directions.

Figure 4.24 Surface Pressure Magnitude and Direction

(a) pressure applied in vertical direction



(b) pressure applied normal to ground surface



For fluid surface pressures, the direction must be specified as normal. The magnitude is computed from the vertical distance between the pressure line and the soil surface. Since the fluid pressure is the same in all directions, the pressure can be applied normal to the surface.

The pressure on top of each slice is multiplied by the surface area and the result is applied as a force on top of each slice.

The Draw Menu

The main function of Draw is to define data by pointing, dragging, and clicking a mouse.

The Draw menu commands are:

- ***Points*** Defines points used in specifying the geometric data. For more information about this command, see Draw Points in this chapter.
- ***Points on Mesh*** Defines points at the same *x-y* coordinates as the nodes in a finite element mesh. For more information about this command, see Draw Points on Mesh in this chapter.
- ***Lines*** Defines the boundaries for each soil. For more information about this command, see Draw Lines in this chapter.
- ***Slip Surface: Grid*** Defines rotation centers for circular and composite slip surfaces. For more information about this command, see Draw Slip Surface: Grid in this chapter.
- ***Slip Surface: Radius*** Defines lines that control the slip surface radii. For more information about this command, see Draw Slip Surface: Radius in this chapter.
- ***Slip Surface: Axis*** Selects the point about which to compute moment equilibrium. For more information about this command, see Draw Slip Surface: Axis in this chapter.
- ***Slip Surface: Specified*** Defines the slip surfaces as piece-wise linear line segments. For more information about this command, see Draw Slip Surface: Specified in this chapter.
- ***Slip Surface: Left Block*** Defines the left block of intersection points for a generated piece-wise linear slip surface. For more information about this command, see Draw Slip Surface: Left Block in this chapter.
- ***Slip Surface: Right Block*** Defines the right block of intersection points for a generated piece-wise linear slip surface. For more information about this command, see Draw Slip Surface: Right Block in this chapter.
- ***Slip Surface: Limits*** Defines the limits within which the slip surface must intersect the top soil layer. For more information about this command, see Draw Slip Surface: Limits in this chapter.
- ***Pore-Water Pressure*** Defines the pore-water pressure conditions. For more information about this command, see Draw Pore Water Pressure in this chapter.
- ***Line Loads*** Sets the position, magnitude, and direction of concentrated loads. For more information about this command, see Draw Line Loads in this chapter.
- ***Anchor Loads*** Defines anchors acting as concentrated loads within the soil. For more information about this command, see Draw Anchor Loads in this chapter.
- ***Pressure Lines*** Defines pressure lines applied at the top soil surface. For more information about this command, see Draw Pressure Lines in this chapter.
- ***Tension Crack Line*** Defines the tension crack line. For more information about this command, see Draw Tension Crack Line in this chapter.

Draw Points

☐ Defines points used in specifying the geometric data.

Points are used to specify line segment endpoints, grid centers, line and anchor load positions, and all other data required by SLOPE/W. Points are created automatically when you use any of the other DEFINE Draw commands if a point does not already exist at the position where you have clicked.

➢ **To draw points:**

1. Choose Points from either the Draw menu or the Mode Toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Points" is the current mode.

2. Move the cursor to the desired position and click the left mouse button.

   A small black square appears at the point position.

3. Repeat Step 2 for all desired points.

4. Click the right mouse button to finish drawing points.

Comments

The point that you define will be placed at a grid point if the Snap to Grid option is on. To toggle the Snap to Grid option, use the Set Grid command or the Snap Grid button on the Grid toolbar.

Points can be moved or deleted using the Modify Objects command. You can also choose KeyIn Points to change the *x-y* coordinates of a point or delete a point. Deleting a point will not delete any lines that may be connected to the point; the point simply will be removed from the line data.

See the KeyIn Points section for information on how points are used to define the geometry, grid of centers, and radius lines.

Draw Points on Mesh

☐ Defines points at the same *x-y* coordinates as the nodes in a finite element mesh.

When the pore-water pressures or stresses from a finite element analysis are to be used in a SLOPE/W stability analysis, it is necessary to define the SLOPE/W data in the same geometric environment as the finite element mesh. Draw Points on Mesh lets you create SLOPE/W points on top of the finite element nodes, so that the geometry can be defined on top of the finite element mesh.

➢ **To draw points on the mesh:**

1. Choose Points on Mesh from either the Draw menu or the Mode Toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Points on Mesh" is the current mode.

2. Move the cursor close to a corner node in the mesh and click the left mouse button.

   The point is drawn as a small black square at the same location as the node. If no point is drawn, move the cursor closer to the corner node and click again.

3. Repeat Step 2 for all desired points.

4. Press ESC or click the right mouse button to finish drawing points on the mesh.

Comments

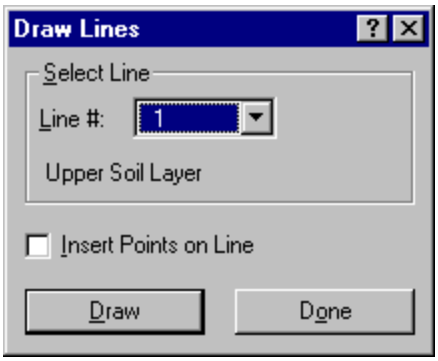
The Draw Points on Mesh command is only available if a mesh has been imported using File Import: Data File.

Draw Lines

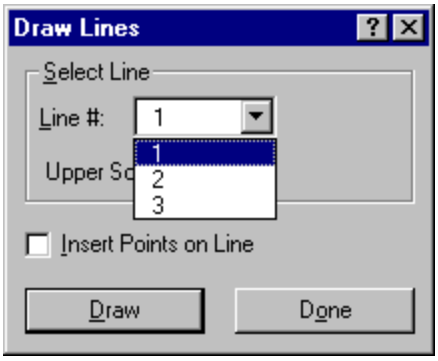
☐ Defines the boundaries for each soil.

➢ **To draw lines:**

1. Choose Lines from either the Draw menu or the Mode Toolbar. The following dialog box appears:



2. Click on the down-arrow to the right of the Line # edit box. A drop-down list box appears, containing one line number for each soil previously defined using KeyIn Soil Properties:



3. Select the line number to draw from the drop-down list box. The corresponding soil description is displayed underneath the selected line number.

4. Select the Draw button.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Lines" is the current mode. If the line to draw already contains points, the line is shaded red and the existing points on the line are highlighted; a line is drawn from the last point in the line to the cursor position.

5. Move the cursor to the left-most position of the line, and click the left mouse button.

   As you move the cursor, a black line appears, indicating the line is being drawn.

   When defining line segments, if you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest grid point.

6. Move the cursor to the position of the next point on the line, and click the left mouse button.

   A red line is drawn from the last point on the line to this point (the point is created if you did not click close to any other point).

7. Repeat Step 6 for all points on the line.

8. Click the right mouse button to finish drawing the line.

   The Draw Lines dialog box reappears.

9. If you wish to draw another line, repeat Steps 2 to 8. Otherwise, select Done in the dialog box to finish drawing lines.

- **To insert points on an existing line:**

1. Choose Draw Lines and select the desired line number.

2. Check the Insert Points on Line check box and select Draw.

   The selected line is shaded red and the existing points on the line are highlighted.

3. Move the cursor to the position on the line at which to insert a point, and click the left mouse button.

   A point is added to the line at the cursor position.

4. Repeat Step 3 for each point to add to the line.

5. Click the right mouse button to finish inserting points in the line.

   Once points have been added to a line, they can be moved by choosing the Move Points command.

- **To define a line for which all of the points are not displayed in the DEFINE window:**

1. Choose Draw Lines and define as much of the line as you can.

2. Click the right mouse button to finish drawing the line.

3. Scroll the drawing to the right.

4. Choose Draw Lines again. A line is drawn from the last point defined in the line to the cursor position.

5. Define the remaining points on the line.

- **To define the remainder of a new line along a previously defined line:**

1. Choose Draw Lines and define the new line until the line is on a point also present in an existing line.

2. Click the right mouse button.

   The new line is extended to the right along the existing line and is displayed as a red line. All points in the existing line to the right of the current point are added to the new line.

Comments

SLOPE/W will create a point if no point exists at the position where you have clicked. If a point exists, then the line will snap to the point.

Each line defines the top boundary of the corresponding soil number. The bottom boundary of the soil is defined by the subsequent line number. Figure 4.25 shows how the line and soil numbers are related.

Each line must start at the left extremity of the problem and extend across the problem to the right extremity. In other words, the starting and ending *x*-coordinates of each line must match the left and right boundaries of the problem.

The geometry lines must be specified in descending order, starting with the ground or water surface and ending with the bottom soil layer. Water impounded against a slope must be defined as Soil 1 (Line 1) or as a surface pressure line.

Lines must be defined from left to right. In other words, the *x*-coordinate of each point in a line must be greater than the *x*-coordinate of the previous point in the line. Vertical segments in a line are not permitted.

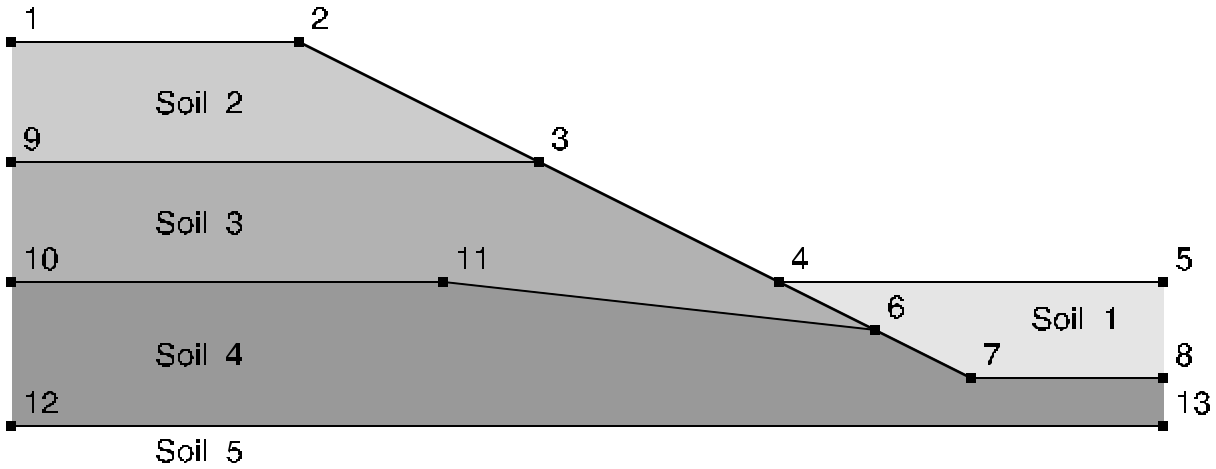
The geometry must be wide enough to include all potential slip surfaces; SOLVE will not compute a factor of safety for any slip surfaces that extend beyond the geometry. When a grid of slip surface centers is used, the left and right boundaries of the geometry should extend beyond the slip circle with the largest radius.

Lines defining discontinuous strata must also extend from the left to the right boundaries of the geometry. Beyond the point at which the strata becomes discontinuous, the top and bottom lines of the layer are superimposed, as shown in Figure 4.26.

Lines must not cross other lines. Two lines can meet and overlap, but they cannot cross.

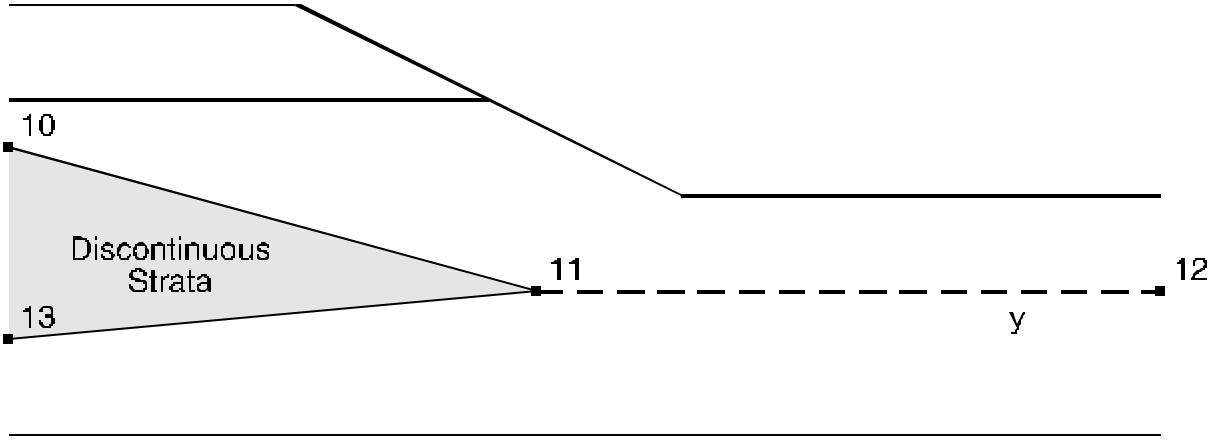
Choose the Tools Verify command to help you verify that the lines have been defined correctly.

Figure 4.25 Definition of Lines



Line 1: Points 1, 2, 3, 4, 5
Line 2: Points 1, 2, 3, 4, 6, 7, 8
Line 3: Points 9, 3, 4, 6, 7, 8
Line 4: Points 10, 11, 6, 7, 8
Line 5: Points 12, 13

Figure 4.26 Definition of a Discontinuous Strata



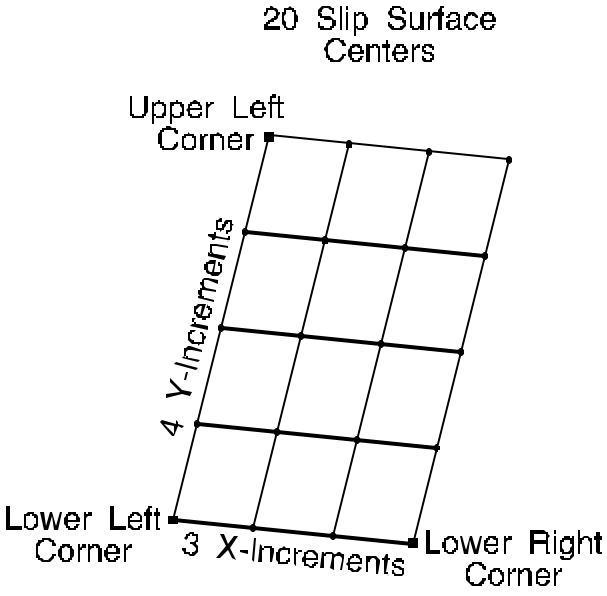
Top Line: Points 10, 11, 12
Bottom Line: Points 13, 11, 12

Draw Slip Surface Grid

☐ Defines the rotation centers for circular and composite slip surfaces.

Figure 4.27 shows the three corner points that define the grid boundaries and the increments that define the number of slip surface centers in the grid.

Figure 4.27 Definition of the Grid of Slip Surface Centers



➢ **To draw the grid:**

1. Choose Slip Surface Grid from either the Draw menu or the Mode Toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that “Draw Slip Surface Grid” is the current mode.

2. Move the cursor near the position of the **upper left** corner of the grid, and click the left mouse button.

   As you move the cursor, a black line appears, indicating how much of the grid has already been defined.

   When defining the grid corners, if you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest grid point.

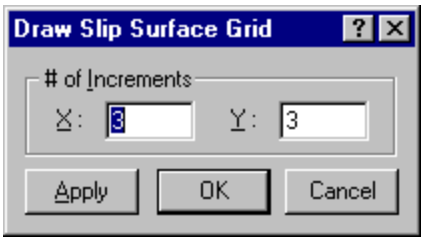
3. Move the cursor near the position of the **lower left** corner of the grid, and click the left mouse button.

   As you move the cursor, a black parallelogram appears, indicating how much of the grid has already been defined. If you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest grid point.

4. Move the cursor near the position of the **lower right** corner of the grid, and click the left mouse button.

   If you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest grid point.

   A parallelogram is drawn connecting the three grid corner points. The following dialog box appears, allowing you to specify the number of increments in which to divide each side of the grid:



   The X Increment edit box is highlighted along with the corresponding *x*-increment side of the parallelogram.

   The total number of slip surface centers in the grid is equal to:

$$N = (X + 1) \times (Y + 1)$$

   where N is the number of slip surface grid centers, X is the number of x-increments, and Y is the number of y-increments.

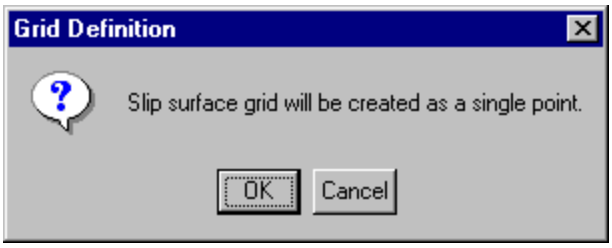
5. Type the number of *x*-increments in the X edit box and the number of *y*-increments in the Y edit box.

6. Select the Apply button to preview the resulting grid of centers.

7. When you are satisfied with the slip surface grid, select OK. Alternatively, you can select Cancel to abort the grid definition; any previously-defined slip surface grid will be restored.

➢ **To define a single slip surface center:**

- Click on the same point for all three grid corner points. The Draw Slip Surface Grid dialog box will not appear, since you do not need to specify any increments. However, the following message appears, allowing you to confirm the single slip surface center:



➢ **To define a series of centers along a non-vertical straight line:**

1. Click on the same point for the upper left and lower left corner points.
2. Click on a different lower right corner point.

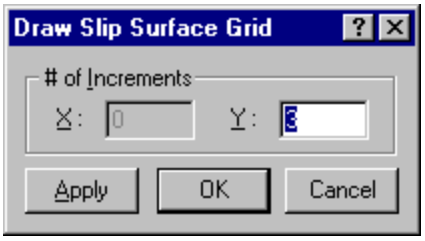
   The Draw Slip Surface Grid dialog box appears, allowing you to specify the number of *x*-increments along the line; the number of *y*-increments is zero:

3. Specify the number of *x*-increments, and select OK to generate a line of slip surface centers.

➢ **To define a series of centers along a non-horizontal straight line:**

1. Click on the upper left corner point
2. Click on the same point for the lower left and lower right corner points.

   The Draw Slip Surface Grid dialog box appears, allowing you to specify the number of *y*-increments along the line; the number of *x*-increments is zero:



3. Specify the number of *y*-increments, and select OK to generate a line of slip surface centers.

Comments

This command can only be chosen if you have selected the Grid & Radius slip surface option with the KeyIn Analysis Control command.

The position and shape of the grid can be modified by moving the grid corner points with the Modify Objects command.

Choose the KeyIn Slip Surface Grid & Radius command to modify the number of increments in each direction. This command also allows you to define projection angles, which are used to specify a straight line segment as the left or right side of the slip surface.

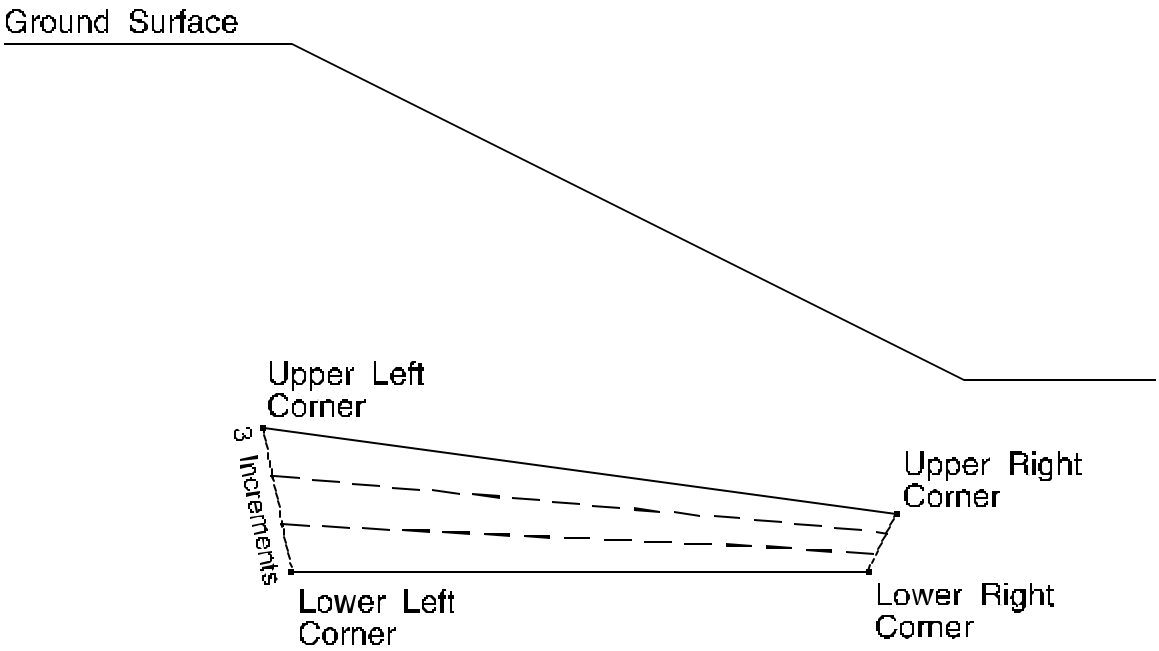
Draw Slip Surface Radius

☐ Defines lines that control the slip surface radii.

The radii of potential slip circles are defined by lines that are tangent to the circles. SLOPE/W computes the radius for each slip circle as the perpendicular distance from each "radius line" to each grid center.

Figure 4.28 shows the four corner points that define the radius line boundaries and the increments that define the number of radius lines to draw.

Figure 4.28 Definition of Four Radius Tangent Lines



➢ **To draw the radius lines:**

1. Choose Slip Surface Radius from either the Draw menu or the Mode Toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that “Draw Slip Surface Radius” is the current mode.

2. Move the cursor near the position of the **upper left** corner of the radius line region, and click the left mouse button.

   As you move the cursor, a black line appears, indicating how much of the radius region has already been defined.

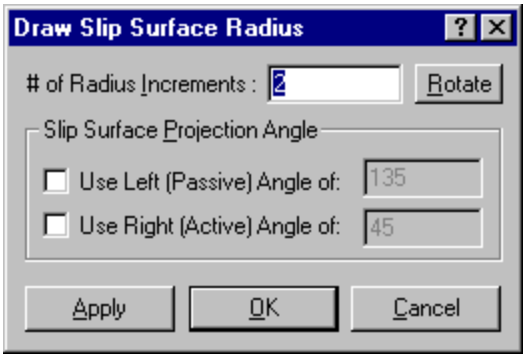
   When defining the radius corners, if you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest grid point.

3. Move the cursor near the position of the **lower left** corner of the radius region, and click the left mouse button.

4. Move the cursor near the position of the **lower right** corner of the radius region, and click the left mouse button.

5. Move the cursor near the position of the **upper right** corner of the radius region, and click the left mouse button.

   A quadrilateral is drawn connecting the four radius corner points. The following dialog box appears, allowing you to specify the number of increments in which to divide each side of the quadrilateral region:



   The # of Radius Increments edit box is highlighted along with the corresponding side of the defined region. The total number of radius lines in the region will be one greater than the number of radius increments.

6. Type the number of radius increments in the edit box.

7. Select the Apply button to preview the resulting radius lines.

8. If the radius lines were generated along the wrong side of the region, select the Rotate button to rotate the radius lines by 90 degrees. Select Rotate again to return the lines to their original position.

9. If you wish to specify a projection angle along the left side of the slip surface, check the Left Angle check box in the Slip Surface Projection Angle group box and specify an angle in the edit box.

   If the left projection angle is on the active (crest) side of the problem (i.e., the slip surface movement is from left to right), the angle must be between 100º and 135º; if the left projection angle is on the passive (toe) side, it must be between 120º and 180º.

10. If you wish to specify a projection angle along the right side of the slip surface, check the Right Angle check box in the Slip Surface Projection Angle group box and specify an angle in the edit box.

    If the right projection angle is on the active (crest) side of the problem (i.e., the slip surface movement is from right to left), the angle must be between 45º and 80º; if the right projection angle is on the passive (toe) side, it must be between 0º and 60º.

NOTE: Slip surface projection angles are used to specify a straight line segment as the left or right side of the slip surface. See KeyIn Slip Surface: Grid & Radius for more information on slip surface projection angles.

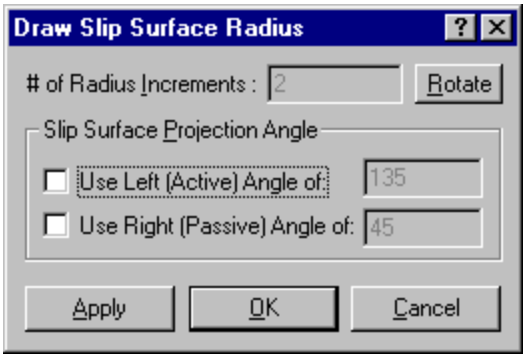
11. Select the Apply button to preview the radius lines.

12. When you are satisfied with the slip surface radius lines, select OK. Alternatively, you can select Cancel to abort the radius line definition; any previously-defined slip surface radius lines will be restored.

➢ **To define a single radius line for each slip surface center:**

1. Click twice on the same left point and then click twice on the same right point.

The Draw Slip Surface Radius dialog box appears, allowing you to specify the slip surface projection angles; the number of radius increments is disabled, since only a single radius line is defined:



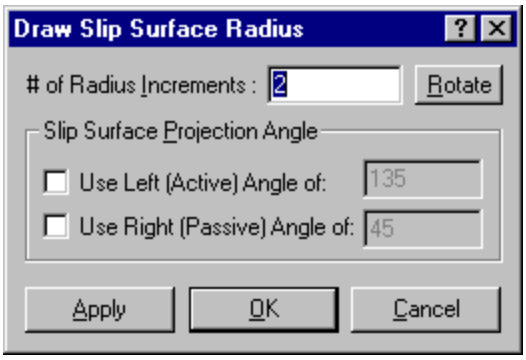
2. Specify the Slip Surface Projection Angles as required, and select OK to define the single radius line.

NOTE: Selecting the Rotate button will toggle the definition between a single radius line and a series of tangent points along a line that the slip surfaces would pass through. If you click the Rotate button, you can specify the # of Radius Increments and select Apply to see the radius tangent points.

➢ **To define a series of slip surface tangent points along a line:**

1. Click on an upper point, then click twice on a lower point, and then click again on the same upper point.

   The Draw Slip Surface Radius dialog box appears:



2. Type the number of radius increments in the edit box, and specify the Slip Surface Projection Angles as required.

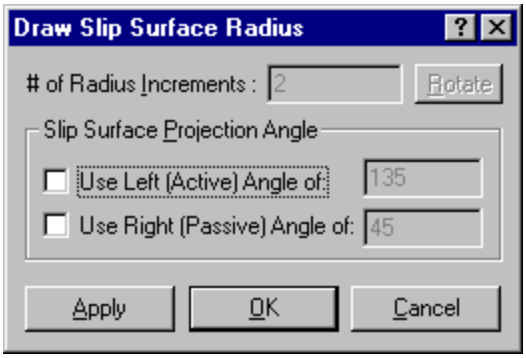
3. Select OK to define the series of slip surface tangent points.

NOTE: Selecting the Rotate button will toggle the definition between a single radius line and a series of tangent points along a line that the slip surfaces would pass through. If you click the Rotate button, the # of Radius Increments will be disabled, since you will be defining a single radius line.

➢ **To force all slip surfaces to pass through a single point:**

1. Click on the same point for all four radius corner points.

The Draw Slip Surface Radius dialog box appears, allowing you to specify the slip surface projection angles; the number of radius increments is disabled, since only a single radius tangent point is defined:



2. Specify the Slip Surface Projection Angles as required, and select OK to define the single radius tangent point.

Comments

This command can only be chosen if you have selected the Grid & Radius slip surface option with the KeyIn Analysis Control command.

The position and shape of the radius lines can be modified by moving the radius corner points with the Modify Objects command.

Choose the KeyIn Slip Surface Grid & Radius command to modify the number of increments in each direction or the slip surface projection angles.

The radius lines do not have to be parallel to each other.

Draw Slip Surface Axis

☐ Selects the point about which to compute moment equilibrium.

The axis point is the point about which moment forces are summed to compute the moment equilibrium factor of safety. The Theory section explains the relationship between the point used to define the circular portion of the slip surface and the moment equilibrium axis.

An axis point must be defined if the Fully Specified or Block Specified slip surface option has been selected using the KeyIn Analysis Control command. The axis point is optional if the Grid & Radius option has been selected.

➢ **To specify an axis point:**

1. Choose Slip Surface Axis from either the Draw menu or the Mode Toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Slip Surface Axis" is the current mode.

2. Move the cursor near the position that will be used for summing the moments, and click the left mouse button.

   If you click close to a point, the cursor snaps to the point; otherwise, an axis point is created at the cursor position. The axis point is displayed as a green dot with a cross-hair.

Comments

The point used for the slip surface axis can be deleted or moved with the Modify Objects command. The KeyIn Slip Surface Axis command allows you to change the axis point number or delete it by setting the axis point number to zero.

The axis point can be deleted by selecting 0 in the Point # drop-down list box or by choosing Modify Objects and deleting the axis point.

Methods that satisfy both moment and force equilibrium (e.g., Morgenstern-Price and GLE) are insensitive to the axis point used to sum moments. Methods that satisfy only moment or only force equilibrium can be slightly affected by the moment equilibrium point (see the Moment Axis section of Chapter 8 for more information). As a general rule, the axis point should be located approximately at the center of rotation of the slip surfaces.

Draw Slip Surface Specified

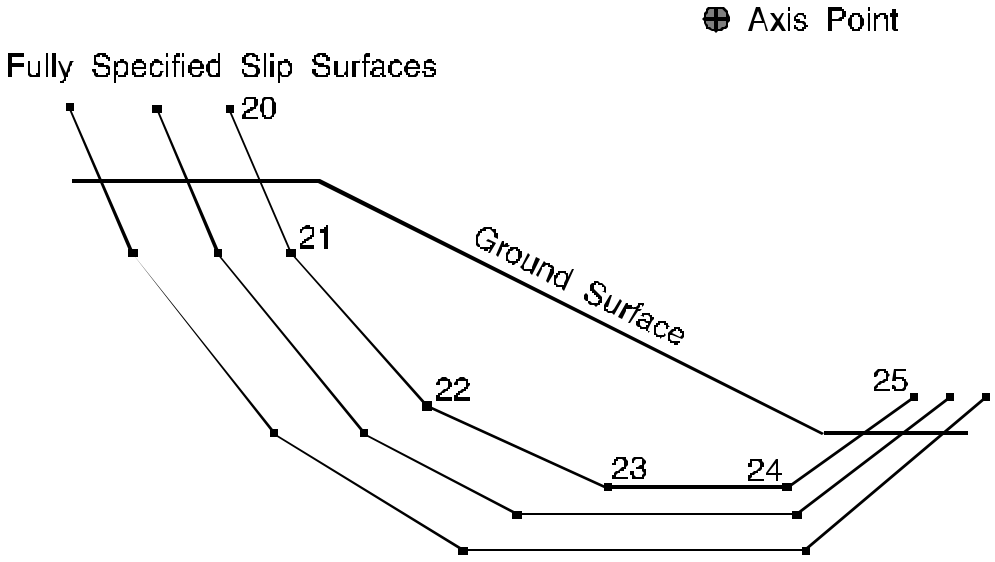
☐ Defines the slip surfaces as piece-wise linear line segments.

Fully specified slip surfaces are slip surfaces made up of a series of line segments. Each slip surface must be specified individually by defining the points that form the line segments for each slip surface line.

This command can only be chosen if you have selected the Fully Specified slip surface option using the KeyIn Analysis Control command.

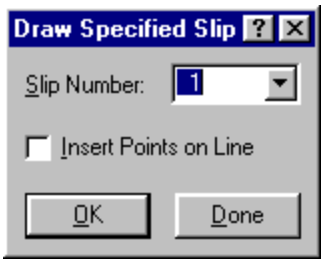
Figure 4.29 shows the definition of three fully specified slip surfaces.

Figure 4.29 Definition of Fully Specified Slip Surfaces



➢ **To draw fully specified slip surfaces:**

1. Choose Slip Surface Specified from either the Draw menu or the Mode Toolbar. The following dialog box appears:



2. In the Slip Number edit box, select or type the slip surface number. If the slip surface already has been defined, click on the down-arrow to display slip surface numbers already specified.

3. Select OK.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Specified Slips" is the current mode. If the slip surface to draw already contains points, the slip surface is shaded red and the existing points on the slip surface are highlighted; a line is drawn from the last point in the slip surface to the cursor position.

4. Move the cursor near the left-most position of the slip surface, and click the left mouse button. The position should be above the crest of the slope, such as Point 20 in Figure 4.29.

   As you move the cursor, a black line appears, indicating the slip surface is being drawn.

   When specifying slip surfaces, if you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest grid point.

5. Move the cursor near the next left-most position to define on the slip surface (such as Point 21 in Figure 4.29), and click the left mouse button.

   A red line is drawn from the last point on the slip surface to this point (the point is created if you did not click close to any other point).

6. Repeat Step 5 for all points to define on the slip surface.

7. Press ESC or click the right mouse button to finish drawing the slip surface.

   The Draw Specified Slip Surfaces dialog box reappears.

8. If you wish to draw another slip surface, repeat Steps 2 to 7. Otherwise, select Done in the dialog box to finish drawing fully specified slip surfaces.

➢ **To insert points on an existing fully specified slip surface:**

1. Choose Slip Surface Specified from either the Draw menu or the Mode Toolbar and select the desired slip surface number.

2. Check the Insert Points on Line check box and select OK.

   The selected slip surface is shaded red and the existing points on the slip surface are highlighted.

3. Move the cursor to the position on the slip surface at which to insert a point, and click the left mouse button.

   A point is added to the slip surface at the cursor position.

4. Repeat Step 3 for each point to add to the slip surface.

5. Press ESC or click the right mouse button to finish inserting points in the slip surface.

   Once points have been added to a slip surface, they can be moved by choosing the Modify Objects command.

Comments

An axis point must be defined using Draw Slip Surface: Axis before any fully specified slip surfaces can be defined. Figure 4.29 above shows the definition of an axis point and three fully specified slip surfaces.

The position and shape of the fully specified slip surface can be modified by moving the slip surface points with the Modify Objects command.

The first and last endpoint of each fully specified slip surface must lie above the top of the geometry (i.e., Soil Line 1). If either endpoint lies underneath Soil Line 1, an error will be displayed when you choose the Tools Verify command.

Fully specified slip surfaces must be defined from left to right. In other words, the *x*-coordinate of each point in a slip surface must be greater than the *x*-coordinate of the previous point in the slip surface. Vertical line segments in a specified slip surface are not permitted.

The left and right endpoints of the fully specified slip surfaces must not extend beyond the boundaries of the geometry lines.

Draw Slip Surface Left Block

☐ Defines the left block of intersection points for a generated piece-wise linear slip surface.

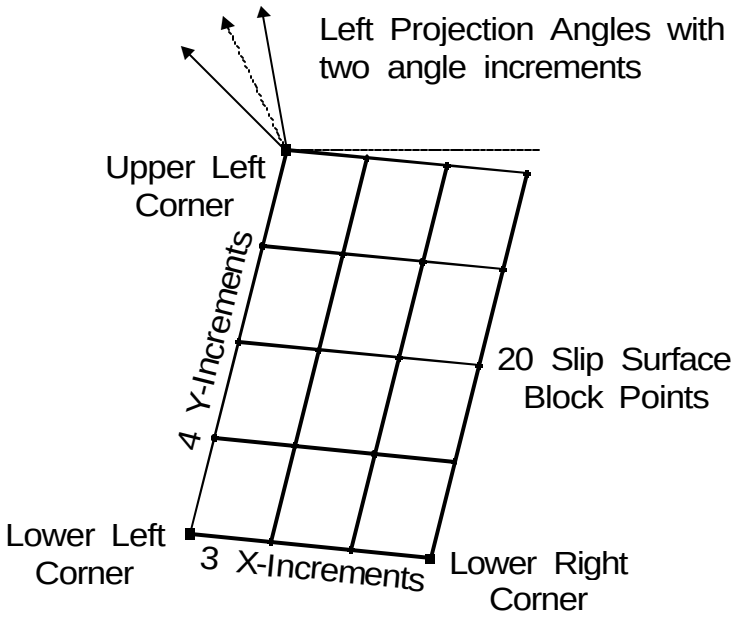
The left slip surface block points are used as the left intersection points of a generated piece-wise linear slip surface. A block-specified slip surface consists of several line segments defined by two grids of intersection points. Slip surfaces are created by connecting each point in the left block with each point in the right block, and then projecting each point to the surface at specified angles.

The surface projection angles are defined by entering a range of angles and the number of increments used to subdivide the range.

This command can only be chosen if you have selected the Block Specified slip surface option with the KeyIn Analysis Control command.

Figure 4.30 shows the position of the three block corner points, the projection angles, and the meaning of the increment values.

Figure 4.30 Definition of the Left Slip Surface Block



➢ **To draw the left slip surface block:**

1. Choose Slip Surface Left Block from either the Draw menu or the Mode Toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Slip Surface Left Block" is the current mode.

2. Move the cursor near the position of the **upper left** corner of the block, and click the left mouse button.

   As you move the cursor, a black line appears, indicating how much of the block has already been defined.

   When defining the block corners, if you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest block point.

3. Move the cursor near the position of the **lower left** corner of the block, and click the left mouse button.

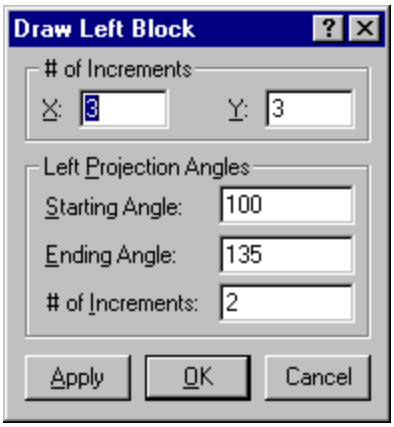
   As you move the cursor, a black parallelogram appears, indicating how much of the block has already been defined. If you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest block point.

4. Move the cursor near the position of the **lower right** corner of the block, and click the left mouse button.

   If you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest block point.

A parallelogram is drawn connecting the three block corner points. Two small arrows are displayed at the upper-left block point to represent the starting and ending left block projection angles. These are the range of angles at which the left portion of the slip surface will be projected to the soil surface.

The following dialog box appears, allowing you to specify the number of increments in which to divide each side of the block:



The X Increment edit box is highlighted along with the corresponding *x*-increment side of the parallelogram.

5. Type the number of *x*-increments in the X edit box and the number of *y*-increments in the Y edit box.

   The total number of slip surface points in the left block grid is equal to:

   $$LB = (X + 1) \times (Y + 1)$$

   where *LB* is the number of slip surface points in the left block, *X* is the number of *x*-increments, and *Y* is the number of *y*-increments. The total number of slip surfaces generated for the problem is equal to:

   $$N = LB \times LA \times RB \times RA$$

   where *N* is the total number of slip surfaces in the problem and *RB* is the number of slip surface points in the right block, *LA* is the number of left block projection angles, and *RA* is the number of right block projection angles.

6. Enter the left block surface projection angle settings in the Left Projection Angles group box.

   Starting Angle Specifies the starting, or minimum, surface projection angle.

   Ending Angle Specifies the ending, or maximum, surface projection angle.

   Number of Increments Specifies the number of increments between the Starting Angle and Ending Angle. The number of projection angles is one greater than the Number of Increments. For example, if 3 increments are specified and the Starting and Ending Angles are 100 and 130 degrees respectively, the resulting projection angles are 100, 110, 120, and 130 degrees. If 0 increments are specified, than the Ending Angle is ignored and only the Starting Angle is used.

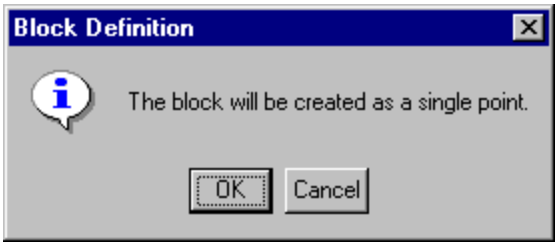
   If the direction of slip surface movement is from left to right, the range of projection angles must be between 100º and 135º; if the direction is from right to left, the range of projection angles must be between 120º and 180º.

7. Select the Apply button to preview the resulting block of points and the starting and ending projection angles.

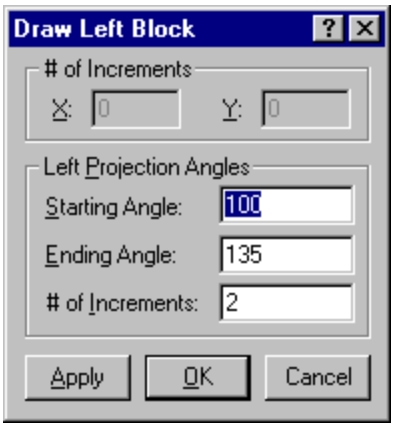
8. When you are satisfied with the slip surface left block, select OK. Alternatively, you can select Cancel to abort the left block definition; any previously-defined slip surface left block will be restored.

➢ **To define a single left block intersection point:**

1. Click on the same point for all three left block corner points. The following message appears, allowing you to confirm the single left block point:



If you select OK, the Draw Slip Surface Left Block dialog box appears, allowing you to specify the left block surface projection angles:



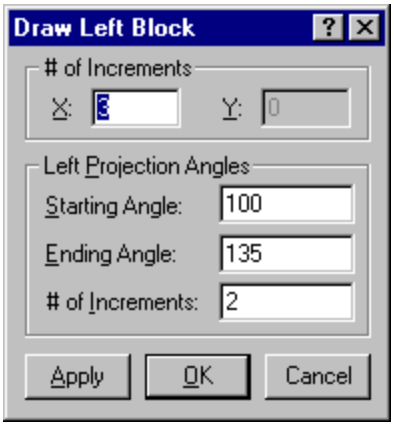
2. Specify the surface projection angle settings, and select OK to create a single left block point.

➢ **To define a series of left block intersection points along a non-vertical straight line:**

1. Click on the same point for the upper left and lower left corner points.

2. Click on a different lower right corner point.

   The Draw Slip Surface Left Block dialog box appears, allowing you to specify the projection angles and the number of *x*-increments along the line; the number of *y*-increments is zero:



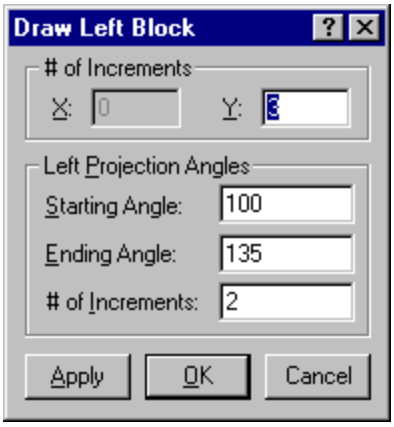
3. Specify the number of *x*-increments, the left projection angle settings, and select OK to generate the line of left block intersection points.

➢ **To define a series of left block intersection points along a non-horizontal straight line:**

1. Click on the upper left corner point

2. Click on the same point for the lower left and lower right corner points.

   The Draw Slip Surface Left Block dialog box appears, allowing you to specify the projection angles and the number of *y*-increments along the line; the number of *x*-increments is zero:



3. Specify the number of *y*-increments, the left projection angle settings, and select OK to generate a line of left block intersection points.

Comments

The position and shape of the block can be modified by moving the block corner points with the Modify Objects command.

Choose the KeyIn Slip Surface Left Block command to modify the number of increments in each direction or to modify the block projection angles.

Choose the KeyIn Slip Surface: Right Block or Draw Slip Surface: Right Block command to specify the right slip surface block of intersection points.

Draw Slip Surface Right Block

☐ Defines the right block of intersection points for a generated piece-wise linear slip surface.

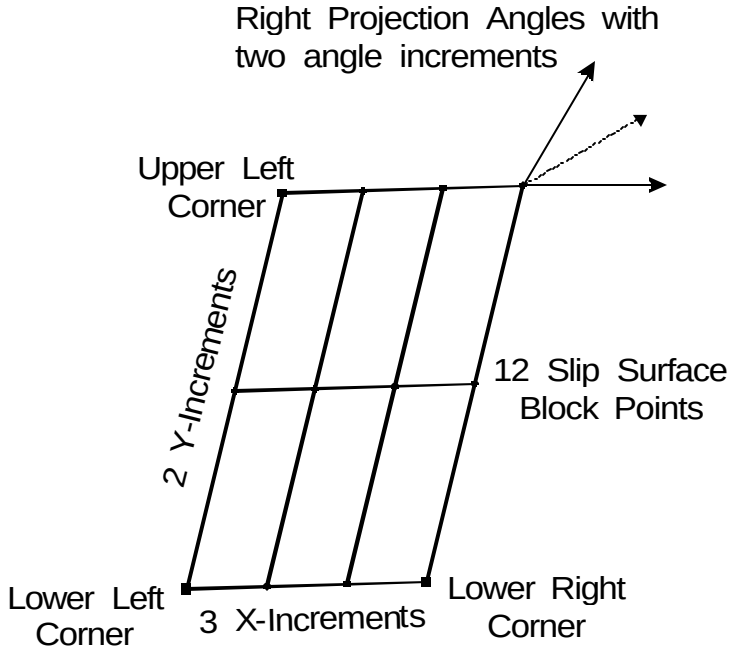
The right block points are used as the right intersection points of a generated piece-wise linear slip surface. A block-specified slip surface consists of several line segments defined by two grids of intersection points. Slip surfaces are created by connecting each point in the left block with each point in the right block, and then projecting each point to the surface at specified angles.

The surface projection angles are defined by entering a range of angles and the number of increments used to subdivide the range.

This command can only be chosen if you have selected the Block Specified slip surface option with the KeyIn Analysis Control command.

Figure 4.31 shows the position of the three block corner points, the projection angles, and the meaning of the increment values.

Figure 4.31 Definition of the Right Slip Surface Block



➢ **To draw the right slip surface block:**

1. Choose Slip Surface Right Block from either the Draw menu or the Mode Toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Slip Surface Right Block" is the current mode.

2. Move the cursor near the position of the **upper left** corner of the block, and click the left mouse button.

   As you move the cursor, a black line appears, indicating how much of the block has already been defined.

   When defining the block corners, if you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest block point.

3. Move the cursor near the position of the **lower left** corner of the block, and click the left mouse button.

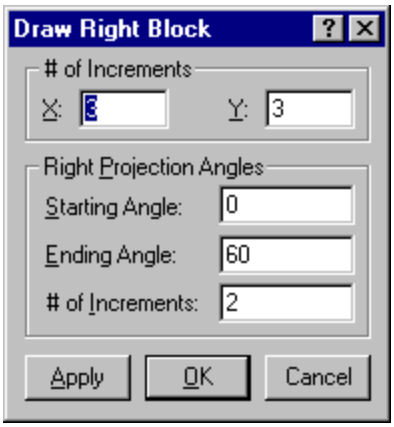
   As you move the cursor, a black parallelogram appears, indicating how much of the block has already been defined. If you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest block point.

4. Move the cursor near the position of the **lower right** corner of the block, and click the left mouse button.

   If you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest block point.

   A parallelogram is drawn connecting the three block corner points. Two small arrows are displayed at the upper-left block point to represent the starting and ending right block projection angles. These are the range of angles at which the right portion of the slip surface will be projected to the soil surface.

   The following dialog box appears, allowing you to specify the number of increments in which to divide each side of the block:



   The X Increment edit box is highlighted along with the corresponding *x*-increment side of the parallelogram.

5. Type the number of *x*-increments in the X edit box and the number of *y*-increments in the Y edit box.

   The total number of slip surface points in the right block grid is equal to:

$$RB = (X + 1) \times (Y + 1)$$

   where *RB* is the number of slip surface points in the right block, *X* is the number of *x*-increments, and *Y* is the number of *y*-increments. The total number of slip surfaces generated for the problem is equal to:

$$N = LB \times LA \times RB \times RA$$

   where *N* is the total number of slip surfaces in the problem and *LB* is the number of slip surface points in the left block, *LA* is the number of left block projection angles, and *RA* is the number of right block projection angles.

6. Enter the right block surface projection angle settings in the Right Projection Angles group box.

   Starting Angle Specifies the starting, or minimum, surface projection angle.

   Ending Angle Specifies the ending, or maximum, surface projection angle.

   Number of Increments Specifies the number of increments between the Starting Angle and Ending Angle. The number of projection angles is one greater than the Number of Increments. For example, if 3 increments are specified and the Starting and Ending Angles are 0 and 60 degrees respectively, the resulting projection angles are 0, 20, 40, and 60 degrees. If 0 increments are specified, than the Ending Angle is ignored and only the Starting Angle is used.

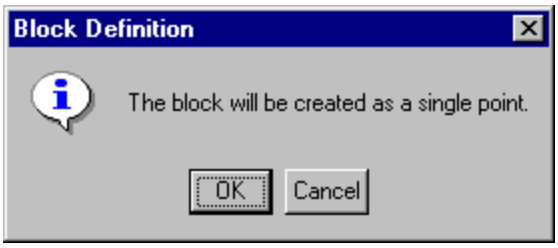
   If the direction of slip surface movement is from left to right, the range of projection angles must be between 0º and 60º; if the direction is from right to left, the range of projection angles must be between 45º and 80º.

7. Select the Apply button to preview the resulting block of points and the starting and ending projection angles.

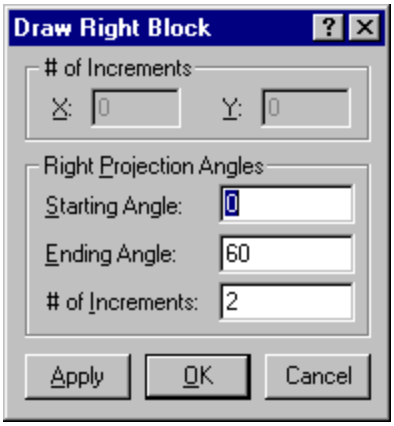
8. When you are satisfied with the slip surface rightblock, select OK. Alternatively, you can select Cancel to abort the right block definition; any previously-defined slip surface right block will be restored.

➢ **To define a single right block intersection point:**

- Click on the same point for all three right block corner points. The following message appears, allowing you to confirm the single right block point:



If you select OK, the Draw Slip Surface Right Block dialog box appears, allowing you to specify the right block surface projection angles:



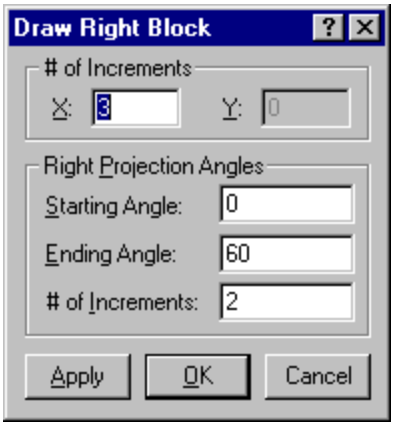
3. Specify the surface projection angle settings, and select OK to create a single right block point.

➢ **To define a series of right block intersection points along a non-vertical straight line:**

1. Click on the same point for the upper left and lower left corner points.

2. Click on a different lower right corner point.

   The Draw Slip Surface Right Block dialog box appears, allowing you to specify the projection angles and the number of *x*-increments along the line; the number of *y*-increments is zero:



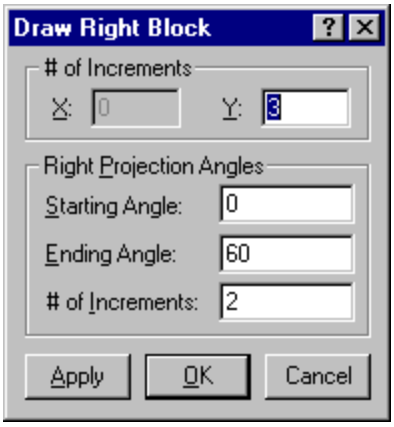
3. Specify the number of *x*-increments, the right projection angle settings, and select OK to generate the line of right block intersection points.

➢ **To define a series of right block intersection points along a non-horizontal straight line:**

1. Click on the upper left corner point

2. Click on the same point for the lower left and lower right corner points.

   The Draw Slip Surface Right Block dialog box appears, allowing you to specify the projection angles and the number of *y*-increments along the line; the number of *x*-increments is zero:



3. Specify the number of *y*-increments, the right projection angle settings, and select OK to generate a line of right block intersection points.

Comments

The position and shape of the block can be modified by moving the block corner points with the Modify Objects command.

Choose the KeyIn Slip Surface: Right Block command to modify the number of increments in each direction or to modify the block projection angles.

Choose the KeyIn Slip Surface: Left Block or Draw Slip Surface: Left Block command to specify the left slip surface block of intersection points.

Draw Slip Surface Limits

☐ Defines the limits within which the slip surface must intersect the top soil layer.

Each trial slip surface must intersect the top soil layer between the left and right limits. SOLVE will not analyze any slip surface that intersects the top soil line beyond these limits. By default, the slip surface limits are set to the left and right edge of the soil geometry. The Draw Slip Surface Limits command allows you to specify limits that are within the soil geometry.

The minimum slip surface limit is displayed above Soil Line 1 as the symbol, . The maximum slip surface limit is displayed above Soil Line 1 as the symbol, . The portion of Line 1 between these two symbols represents the region on the drawing where each potential slip surface is allowed to intersect Line 1.

➢ **To draw the slip surface limits:**

1. Choose Slip Surface Limits either from the Draw menu or the Mode Toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Slip Surface Limits" is the current mode.

2. If you wish to change the left slip surface limit, click on the minimum slip surface limit symbol, , hold down the left mouse button, and drag the symbol along the top soil line (Line 1) to the new minimum *x*-coordinate.

3. If you wish to change the right slip surface limit, click on the maximum slip surface limit symbol, , hold down the left mouse button, and drag the symbol along the top soil line (Line 1) to the new maximum *x*-coordinate.

4. Repeat Steps 2 to 3 until you are finished drawing the slip surface limits.

Comments

If you wish to reset the slip surface limits to be the extents of Soil Line 1, choose KeyIn Slip Surface Limits and select the Line 1 Extents option.

Draw Pore-Water Pressure

☐ Defines the pore-water pressure conditions.

Draw Pore-Water Pressure defines the pore-water pressure according to the method selected with the KeyIn Analysis Control command. Depending on which method has been selected, SLOPE/W obtains the pore-water pressure data for one of the following methods:

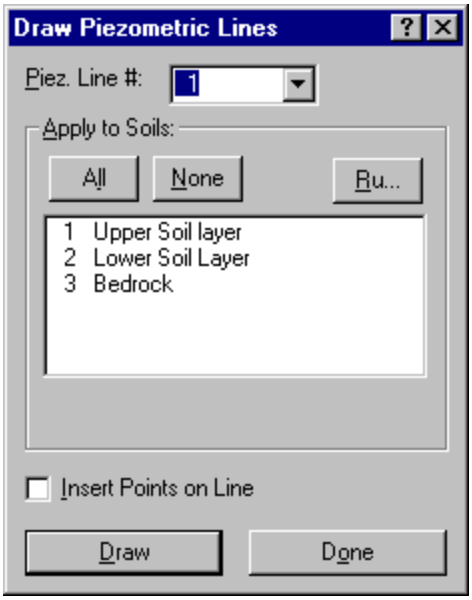
Ru Coefficients

The Draw Pore-Water Pressure command is unavailable if the selected P.W.P. option is R_u Coefficients. The r_u coefficients must be specified for each soil by choosing KeyIn Pore Pressure: Water Pressure. See the KeyIn Analysis Control command in this chapter for further discussion on r_u coefficients.

Piezometric Lines / Ru

➢ **To draw piezometric lines:**

1. Choose Pore-Water Pressure either from the Draw menu or the Mode Toolbar. The following dialog box appears:



2. In the Piez. Line # edit box, type the piezometric line number to define. A list of piezometric lines already defined can be obtained by clicking the arrow to the right of the edit box. Select one of these numbers if you wish to modify a piezometric line that has already been defined.

   If the piezometric line has already been defined, the soils applied to the piezometric line will be selected in the Apply To Soils list box.

3. In the Apply To Soils list box, select the soils to apply to the piezometric line by clicking on each soil in the list box. Click on the soil again to unselect it. Select All to apply all soils to the piezometric line, or select None to unselect all soils in the list box.

4. Select the Draw button.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw P.W.P" is the current mode. If the piezometric line to draw already contains points, a line is drawn from the last point in the piezometric line to the cursor position.

5. Move the cursor near the next left-most position to define on the piezometric line, and click the left mouse button.

   When specifying piezometric lines, if you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest grid point.

   As you move the cursor, a black dashed line appears, indicating the piezometric line is being drawn.

6. Move the cursor near the next left-most position to define on the piezometric line, and click the left mouse button.

   A red dashed line is drawn from the last point on the piezometric line to this point (the point is created if you did not click close to any other point).

7. Repeat Step 6 for all points to define on the piezometric line.

8. Press ESC or click the right mouse button to finish drawing the piezometric line.

   The Draw Piezometric Lines dialog box reappears.

9. If you wish to draw another piezometric line, repeat Steps 2 to 8. Otherwise, select Done in the dialog box to finish drawing piezometric lines.

   The piezometric line will be deleted when you select Done if no soils are applied to the line.

➢ **To insert points on an existing line:**

1. Choose Draw Pore-Water Pressure and select the desired Piezometric line number.

2. Check the Insert Points on Line check box and select Draw.

   The selected line is shaded red and the existing points on the line are highlighted.

3. Move the cursor to the position on the line at which to insert a point, and click the left mouse button.

   A point is added to the line at the cursor position.

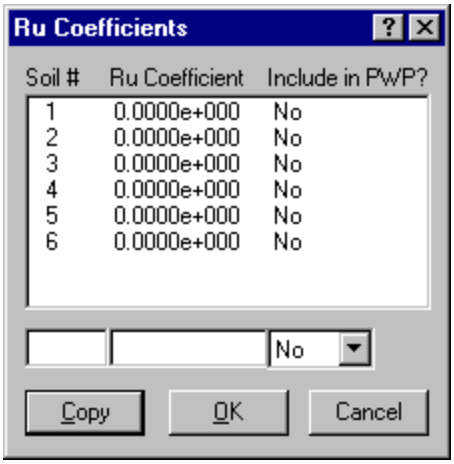
4. Repeat Step 3 for each point to add to the line.

5. Press ESC or click the right mouse button to finish inserting points in the line.

To delete or move points graphically, choose the Modify Objects command. Deleting points will remove them from the piezometric line. Moving points will also move the piezometric lines attached to the points.

➢ **To include the R_u coefficients with the pore-water pressure calculations:**

1. Select the Ru button in the Piezometric Lines dialog box. The following dialog box appears:



2. Select the desired soil number by clicking on the soil in the list box.

   The soil number and its R_u coefficient are copied into the edit boxes. If the soil is to be included in the P.W.P. calculations, Yes is selected in the drop-down list box; otherwise, No is selected.

3. Enter the R_u coefficient value in the R_u Coefficient edit box.

4. Include the soil R_u coefficient in the P.W.P. calculations by selecting Yes in the drop-down list box. To remove the R_u coefficient from the calculations, select No.

5. Select Copy to transfer the data to the list box.

6. Repeat Steps 2 to 5 for all desired soils.

7. Select OK.

   An asterix appears in the Piezometric Lines dialog box beside all soils that have R_u coefficients included in the P.W.P. calculations.

Each piezometric line must start at the left extremity of the problem and extend across the problem to the right extremity. In other words, the starting and ending *x*-coordinates of each piezometric line must match the left and right boundaries of the problem. The Tools Verify command enforces this rule by modifying the *x*-coordinates of the line endpoints if necessary.

Piezometric lines must be defined from left to right. In other words, the *x*-coordinate of each point in a line must be greater than the *x*-coordinate of the previous point in the line. Vertical segments in a line are not permitted.

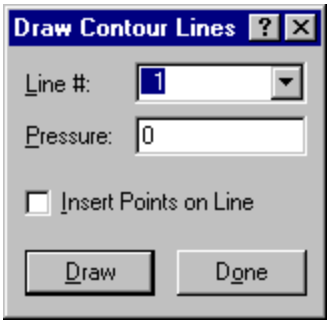
Piezometric lines are displayed on the drawing as blue dashed lines.

See the KeyIn Analysis Control command in this chapter for further discussion on piezometric lines.

Contours

➢ **To draw pressure contour lines:**

1. Choose Pore-Water Pressure either from the Draw menu or the Mode Toolbar. The following dialog box appears:



2. In the Line # edit box, select or type the contour line number to define. If the contour line already has been defined, click on the down-arrow to display the contour line numbers already specified.

3. In the Pressure edit box, type the pore-water pressure value along the contour.

4. Select the Draw button.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw P.W.P" is the current mode. If the contour line to draw already contains points, a line is drawn from the last point in the contour line to the cursor position.

5. Move the cursor near the next left-most position to define on the contour line, and click the left mouse button.

   When specifying contours, if you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest grid point.

As you move the cursor, a black dashed line appears, indicating the contour line is being drawn.

6. Move the cursor near the next left-most position to define on the contour line, and click the left mouse button.

   A red dashed line is drawn from the last point on the contour line to this point (the point is created if you did not click close to any other point).

7. Repeat Step 6 for all points to define on the contour line.

8. Press ESC or click the right mouse button to finish drawing the contour line.

   The Draw Contour Lines dialog box reappears.

9. If you wish to draw another contour line, repeat Steps 2 to 8. Otherwise, select Done in the dialog box to finish drawing pressure contour lines.

➢ **To insert points on an existing pressure contour line:**

1. Choose Draw Pore-Water Pressure and select the desired Contour line number.

2. Check the Insert Points on Line check box and select Draw.

   The selected line is shaded red and the existing points on the line are highlighted.

3. Move the cursor to the position on the line at which to insert a point, and click the left mouse button.

   A point is added to the line at the cursor position.

4. Repeat Step 3 for each point to add to the line.

5. Press ESC or click the right mouse button to finish inserting points in the line.

To delete or move points graphically, choose the Modify Objects command. Deleting points will remove them from the contour line. Moving points will also move the contour lines attached to the points.

Each contour line must start at the left extremity of the problem and extend across the problem to the right extremity. In other words, the starting and ending *x*-coordinates of each contour line must match the left and right boundaries of the problem. The Tools Verify command enforces this rule by modifying the *x*-coordinates of the line endpoints if necessary.

Contour lines must be defined from left to right. In other words, the *x*-coordinate of each point in a line must be greater than the *x*-coordinate of the previous point in the line. Vertical segments in a line are not permitted.

Contour lines are displayed on the drawing as blue dashed lines.

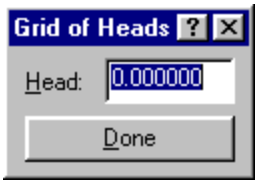
See the KeyIn Analysis Control command in this chapter for further discussion on pore-water pressure contour lines.

Grid of Heads, Grid of Pressures, and Grid of Ru Coefficients

Pressure heads, pressures, and r_u coefficients may be defined at discrete points.

➢ **To define a grid of heads, pressures, or r_u coefficients:**

1. Choose Pore-Water Pressure either from the Draw menu or the Mode Toolbar. The following dialog box appears when defining a grid of heads:



   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw P.W.P" is the current mode.

2. Type the pressure head, pressure, or r_u coefficient in the edit box.

3. Move the cursor near the position at which to define pore-water pressure, and click the left mouse button.

   If you clicked close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest grid point. The point is displayed as a blue triangle.

4. Repeat Step 3 for all points with the same magnitude of pressure, head, or r_u coefficient.

5. If you wish to modify the magnitude, type a new value in the edit box and repeat Step 3 for all points with the new magnitude.

6. Select Done in the dialog box, press ESC, or click the right mouse button to finish defining pore-water pressure grid points.

Pressure head must be defined in the units of length used in the problem, such as metres or feet.

See the KeyIn Analysis Control command in this chapter for further discussion on the grid of heads, grid of pressures, and grid of r_u coefficients.

SEEP/W Heads and SIGMA/W Pressures

The Draw Pore-Water Pressure command is disabled if the selected P.W.P. option is SEEP/W Heads or SIGMA/W Pressures, since the pore-water pressure data is obtained directly from the finite element analysis. See the KeyIn Analysis Control command in this chapter for further discussion on using finite element-computed pore-water pressures.

Draw Line Loads

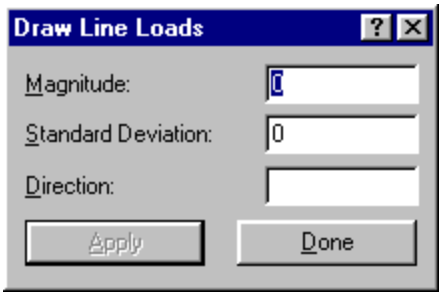
☐ Sets the position, magnitude, and direction of concentrated loads.

Concentrated line loads can exist at points. The loads are defined by the location of the point, the magnitude, and the direction of the load.

Line loads can be used to simulate any concentrated load, such as a structural load or the resultant earth pressure on a retaining wall.

➢ **To draw a line load:**

1. Choose Line Loads from either the Draw menu or the Mode Toolbar. The following dialog box appears:



   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Line Loads" is the current mode.

2. In the Magnitude edit box, type the force value of the load.

3. If you have selected a probability analysis with KeyIn Analysis Control, type the standard deviation of the line load in the Standard Deviation edit box.

4. Move the cursor near the position at which to define the line load, and click the left mouse button.

   When drawing line loads, if you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest grid point.

   As you move the cursor, a black line is drawn from the point to the cursor position. The angle of the line is displayed in the Direction edit box.

5. Move the cursor around the point until the desired angle is displayed in the Direction edit box, and click the left mouse button.

   A small arrow is displayed at the point, pointing in the direction of the line load.

6. To specify an exact angle, type the angle (in degrees) in the Direction edit box.

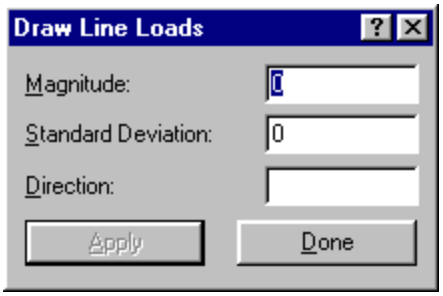
7. Select Apply when you are satisfied with the line load settings.

8. Repeat Steps 2 to 7 for all points at which to define a load.

9. Select Done in the dialog box, press ESC, or click the right mouse button to finish defining line loads.

➢ **To modify an existing line load:**

1. Choose Line Loads from either the Draw menu or the Mode Toolbar. The following dialog box appears:



The cursor changes from an arrow to a cross-hair and the status bar indicates that “Draw Line Loads” is the current mode.

2. Click on a previously-defined line load.

   The selected line load is shown in red. The line load settings are displayed in the Draw Line Loads dialog box.

3. Change the line load settings as necessary.

4. Press the Apply button to apply the new settings to the line load.

5. Repeat Steps 2 to 4 for all line loads that you wish to modify.

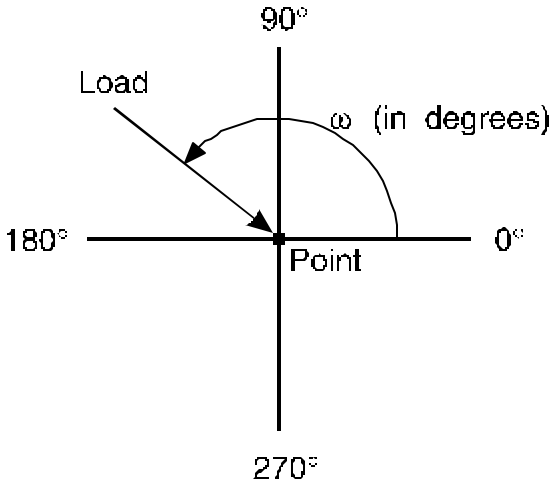
6. Select Done in the dialog box, press ESC, or click the right mouse button to finish modifying line loads.

The direction and magnitude of the line loads also can be modified by choosing KeyIn Load: Line Loads.

To delete or move line loads graphically, choose the Modify Objects command. Deleting the point defining the line load will remove the line load. Moving the line load point will also move the line load.

Figure 4.32 shows the sign convention for line loads.

Figure 4.32 Definition of Line Load Direction



Comments

Line loads affect the forces on the slice to which the load is applied. Both the normal at the base of the slice and forces between adjacent slices are affected.

When you are evaluating the detailed forces on a slice, be aware that more than one line load may affect a particular slice.

Line loads must be applied within the sliding mass to be included in the stability calculations. Line loads outside the sliding mass are ignored.

Draw Anchor Loads

☐ Defines anchors acting as concentrated loads within the soil.

Anchor loads are similar to concentrated line loads (see Draw Line Loads). Using anchor loads, however, allows you to make the load magnitude vary depending on where the slip surface intersects the anchor. Another difference is that anchor loads act at the slice base, whereas line loads act at the point where they are defined.

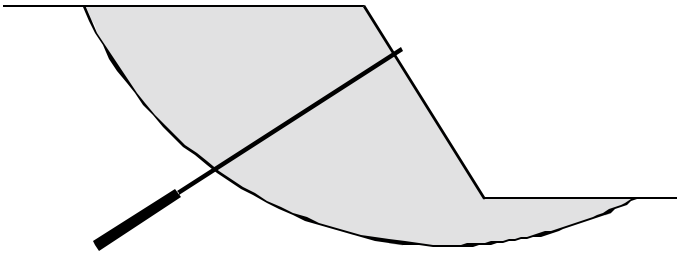
Anchor loads are useful for modelling features such as ground anchors, soil nails, or geofabric reinforcement.

Anchor loads are defined by specifying two points that form a line segment. The first point is defined above the geometry at the position where the anchor is inserted. The second point is defined inside the soil strata at the end of the anchor. A portion of the anchor line is designated as the *bonded length*, representing the part of the anchor that is bonded to the soil.

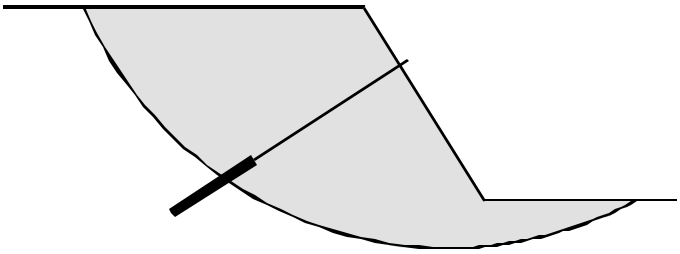
SLOPE/W applies the entire specified anchor load when a slip surface intersects the unbonded portion of the anchor. If the anchor lies completely within the sliding mass, it is ignored. When the slip surface intersects the bonded portion of the anchor, the anchor is ignored if the anchor load is being applied as a constant. If it is being applied variably, then a percentage of the specified magnitude is applied, as shown in Figure 4.33 and 4.34.

Figure 4.33 Definition of Anchor Loads when Applied with Constant Magnitude

(a) Full anchor load is applied



(b) Anchor load is ignored



(c) Anchor load is ignored

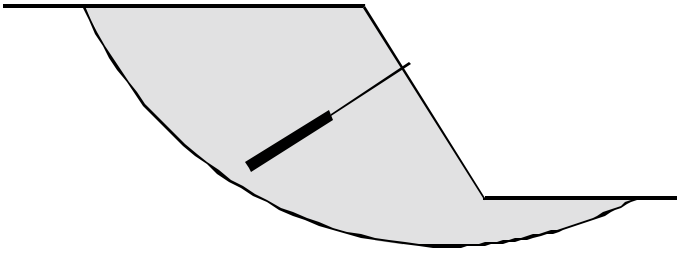
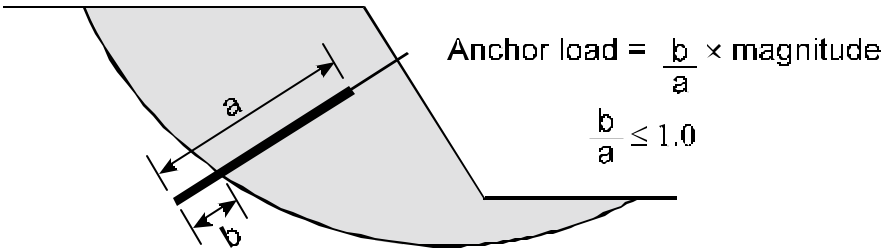
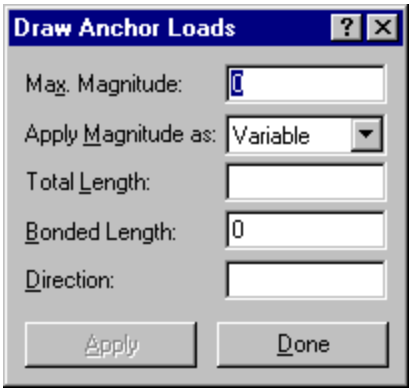


Figure 4.34 Definition of Anchor Loads when Applied with Variable Magnitude



➢ **To draw an anchor load:**

1. Choose Anchor Loads either from the Draw menu or the Mode Toolbar. The following dialog box appears:



   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Anchor Loads" is the current mode.

2. Move the cursor near the position outside the top soil line at which to define the starting anchor point, and click the left mouse button.

   When drawing anchor loads, if you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest grid point.

   As you move the cursor, the Total Length and Direction edit boxes are updated. Also, a black line is drawn from the point to the cursor position.

3. Move the cursor inside the soil geometry near the position at which to define the end of the anchor, and click the left mouse button.

   A line is drawn between the two specified points to indicate the anchor. A small arrow is displayed at the first anchor point, pointing in the direction of the anchor load. The anchor is shown as red to indicate that it is selected.

4. In the Max. Magnitude edit box, type the magnitude of the maximum force applied by the anchor load.

5. Select Constant in the Apply Magnitude as drop-down list box if you do not want any load applied when the slip surface intersects the bonded portion of the anchor. Select Variable if you wish to apply a variable percentage of the load along the bonded portion of the anchor.

6. In the Bonded Length edit box, type the bonded length of the anchor. Press Apply to preview the bonded length relative to the entire anchor length.

   The bonded portion of the anchor is displayed as a thick line segment on the anchor line.

7. Press the Apply button to display the bonded length relative to the entire anchor length.

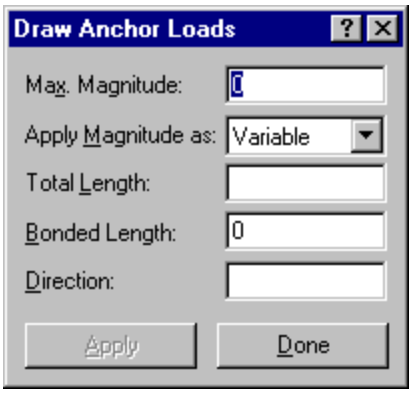
8. If you wish to change the Total Length or Direction of the anchor, enter new values in the corresponding edit boxes, and press the Apply button to apply the changes to the anchor.

   The anchor is drawn at the new Total Length value and is rotated around the first anchor point to the new Direction value.

9. Select Done when you are finished defining the anchor.

➢ **To modify an existing anchor load:**

1. Choose Anchor Loads from either the Draw menu or the Mode Toolbar. The following dialog box appears:



   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Anchor Loads" is the current mode.

2. Click on a previously-defined anchor load.

   The selected anchor load is shown in red. The anchor load settings are displayed in the Draw Anchor Loads dialog box.

3. Change the anchor load settings as necessary.

4. Press the Apply button to apply the new settings to the anchor load.

5. Repeat Steps 2 to 4 for all anchor loads that you wish to modify.

6. Select Done in the dialog box, press ESC, or click the right mouse button to finish modifying anchor loads.

Anchor loads also can be modified by choosing KeyIn Load: Anchor Loads.

To delete or move anchor loads graphically, choose the Modify Objects command. Deleting a point that defines an anchor load will remove the anchor load. Moving an anchor load point will also move the anchor load.

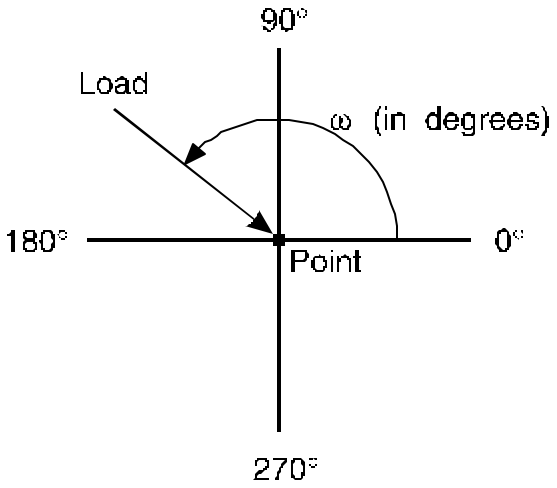
Comments

Anchor loads affect the forces on the slice to which the load is applied. Both the normal at the base of the slice and forces between adjacent slices are affected.

The anchor load is applied to the slice base that intersects the anchor line of action.

Figure 4.32 shows the sign convention for the anchor load direction.

Figure 4.35 Definition of Anchor Load Direction



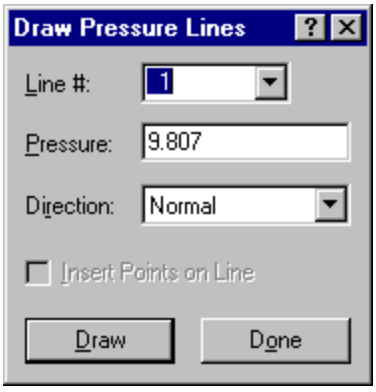
Draw Pressure Lines

☐ Defines pressure lines applied at the top soil surface.

Pressure lines are used to simulate a pressure applied over a portion of the soil surface (e.g., to model a footing on the ground surface). Unlike line loads, which are a concentrated force applied at one point, pressure lines are applied over a region. The magnitude of the applied pressure is computed by multiplying the specified pressure by the vertical distance between the pressure line and the soil surface. The direction of the applied pressure can be specified as normal to the ground surface or vertical.

➢ **To draw pressure lines:**

1. Choose Pressure Lines from either the Draw menu or the Mode Toolbar. The following dialog box appears:



2. Select the pressure line number to draw from the Line # drop-down list box or type a new pressure line number to define.

3. In the Pressure edit box, type the amount of pressure to apply on the portion of the soil surface lying directly underneath the pressure line.

4. In the Direction drop-down list box, select Normal if you wish to apply the pressure at an angle normal to the soil surface. Select Vertical to always apply the pressure vertically down on the soil surface.

5. Select the Draw button.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Pressure Lines" is the current mode. If the pressure line to draw already contains points, the pressure line is shaded red and the existing points on the pressure line are highlighted; a pressure line is drawn from the last point in the pressure line to the cursor position.

6. Move the cursor above the soil geometry to the left-most position of the pressure line, and click the left mouse button.

   As you move the cursor, a black pressure line appears, indicating the pressure line is being drawn.

   When defining pressure line segments, if you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest grid point.

7. Move the cursor to the position of the next point on the pressure line, and click the left mouse button.

   A red pressure line is drawn from the last point on the pressure line to this point (the point is created if you did not click close to any other point). The area underneath the pressure line and above Soil Line 1 is shaded with a cross-hatch pattern.

8. Repeat Step 7 for all points on the pressure line.

9. Press ESC or click the right mouse button to finish drawing the pressure line.

   The Draw Pressure Lines dialog box reappears.

10. If you wish to draw another pressure line, repeat Steps 2 to 9. Otherwise, select Done in the dialog box to finish drawing pressure lines.

- **To insert points on an existing pressure line:**

1. Choose Draw Pressure Lines and select the desired pressure line number.

2. Check the Insert Points on Line check box and select Draw.

   The selected pressure line is shaded red and the existing points on the pressure line are highlighted.

3. Move the cursor to the position on the pressure line at which to insert a point, and click the left mouse button.

   A point is added to the pressure line at the cursor position.

4. Repeat Step 3 for each point to add to the pressure line.

5. Press ESC or click the right mouse button to finish inserting points in the pressure line.

Surface pressure lines also can be modified by choosing KeyIn Pressure Lines.

To delete or move surface pressure line points graphically, choose the Modify Objects command. Deleting one of the points defining the surface pressure line will remove the pressure line. Moving a pressure line point will also move the pressure line.

Comments

SLOPE/W will create a point if no point exists at the position where you have clicked. If a point exists, then the pressure line will snap to the point.

The area underneath each pressure line and above Soil Line 1 is shaded with a cross-hatch pattern to indicate where the surface pressure will be applied on Soil Line 1.

Pressure lines must be defined from left to right. In other words, the *x*-coordinate of each point in a line must be greater than the *x*-coordinate of the previous point in the line. Vertical segments in a line are not permitted.

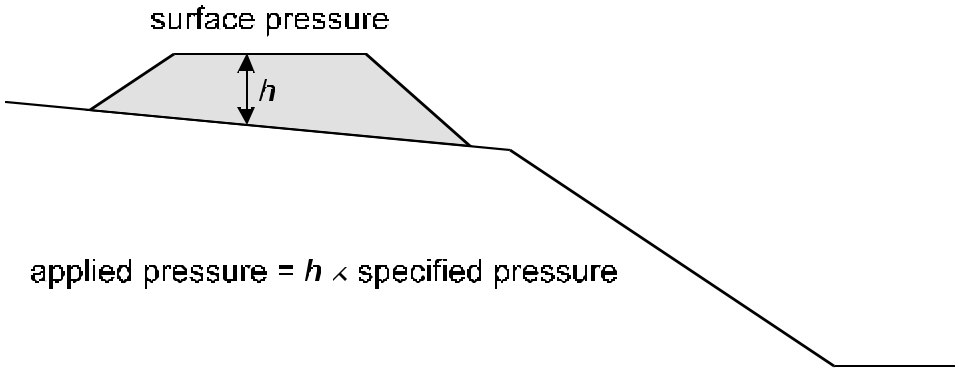
Pressure lines must not cross or overlap with other pressure lines. All pressure lines must lie above the top soil line (Soil Line 1).

Choose the Tools Verify command to help you verify that the surface pressure lines have been defined correctly.

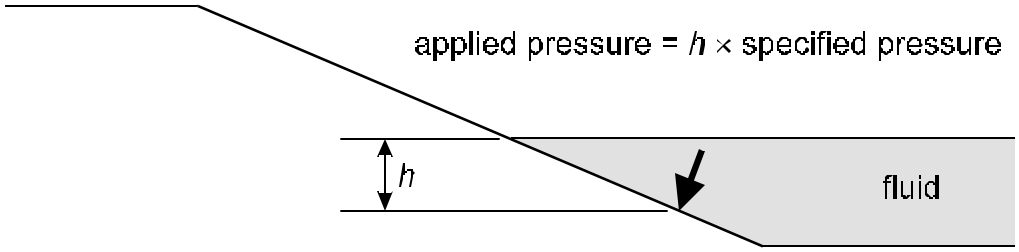
Figure 4.36 illustrates how surface pressures are applied in the normal and vertical directions.

Figure 4.36 Surface Pressure Magnitude and Direction

(a) pressure applied in vertical direction



(b) pressure applied normal to ground surface



For fluid surface pressures, the direction must be specified as normal. The magnitude is computed from the vertical distance between the pressure line and the soil surface. Since the fluid pressure is the same in all directions, the pressure can be applied normal to the surface.

The pressure on top of each slice is multiplied by the surface area and the result is applied as a force on top of each slice.

NOTE: Pressure lines CANNOT be used to specify lateral pressures arising from retaining walls since the magnitude is computed from the vertical distance between the pressure line and the slope surface.

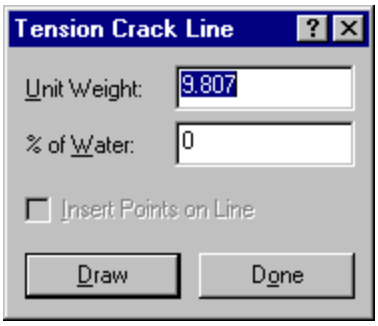
Draw Tension Crack Line

☐ Defines the tension crack line.

If the Tension Crack Line option was selected in KeyIn Analysis Control, the tension crack is modelled by specifying a line across the geometry. The slip surface is projected vertically to the top soil surface at the point where the slip surface intersects the tension crack line.

➢ **To draw the tension crack line:**

1. Choose Tension Crack Line from either the Draw menu or the Mode Toolbar. The following dialog box appears:



2. In the Unit Weight edit box, type the unit weight value of the fluid in the tension crack.

3. In the % of Water edit box, type the percentage of water (from 0to 1) in the tension crack. A value of 0.0 represents a dry tension crack, while a value of 1.0 represents a completely water-filled tension crack.

4. Select the Draw button.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Tension Crack Line" is the current mode. If the tension crack line already contains points, it is shaded red and the existing points on the tension crack line are highlighted; a line is drawn from the last point in the tension crack line to the cursor position.

5. Move the cursor to the left-most position of the tension crack line, and click the left mouse button.

   As you move the cursor, a black line appears, indicating the tension crack line is being drawn.

   When defining tension crack line segments, if you click close to a point, the cursor snaps to the point; otherwise, a point is created at the cursor position or the nearest grid point.

6. Move the cursor to the position of the next point on the tension crack line, and click the left mouse button.

   A red tension crack line is drawn from the last point on the tension crack line to this point (the point is created if you did not click close to any other point). The area above the tension crack line and underneath Soil Line 1 is shaded with a vertical-line pattern.

7. Repeat Step 6 for all points on the tension crack line.

8. Press ESC or click the right mouse button to finish drawing the tension crack line.

   The Draw Tension Crack lines dialog box reappears.

9. If you wish to add more points to the tension crack line, repeat Steps 2 to 8. Otherwise, select Done in the dialog box to finish drawing the tension crack line.

➢ **To insert points on an existing tension crack line:**

1. Choose Draw Tension Crack Lines and Check the Insert Points on Line check box.
2. Select Draw.

   The tension crack line is shaded red and the existing points on the tension crack line are highlighted.
3. Move the cursor to the position on the tension crack line at which to insert a point, and click the left mouse button.

   A point is added to the tension crack line at the cursor position.
4. Repeat Step 3 for each point to add to the tension crack line.
5. Press ESC or click the right mouse button to finish inserting points in the tension crack line.

   Once points have been added to a tension crack line, they can be moved by choosing the Move Points command.

The tension crack line also can be modified by choosing KeyIn Tension Crack.

To delete or move the tension crack line points graphically, choose the Modify Objects command. Deleting one of the points defining the tension crack line will remove the line. Moving a tension crack line point will also move the line.

Comments

SLOPE/W will create a point if no point exists at the position where you have clicked. If a point exists, then the tension crack line will snap to the point.

The tension crack line must start at the left extremity of the problem and extend across the problem to the right extremity. In other words, the starting and ending *x*-coordinates of the tension crack line must match the left and right boundaries of the problem.

Choose the Tools Verify command to help you verify that the tension crack line has been defined correctly.

The tension crack line must be defined from left to right. In other words, the *x*-coordinate of each point in the line must be greater than the *x*-coordinate of the previous point in the line. Vertical line segments in a tension crack line are not permitted.

The tension crack line cannot lie above the top soil surface or within a soil that uses the No Strength soil model (that is, c=0 and $f = 0$).

When the unit weight is greater than zero, SLOPE/W applies a hydrostatic horizontal force on the side of the tension crack. The magnitude of the hydrostatic force is defined as,

$$F = \frac{g\, d^2}{2} \tag{4.3}$$

where:

g = the unit weight of the fluid in the tension crack

d = the depth of the tension crack

The tension crack force is applied at one-third of the depth from the bottom of the crack.

The Sketch Menu

The main function of Sketch is:

- To label, enhance, and clarify the problem definition.
- To create graphic objects which can be used as guide lines for developing the soil geometry.

The Sketch menu commands are:

- ***Lines*** Sketches straight lines. For more information about this command, see Sketch Lines in this chapter.
- ***Circles*** Sketches circles. For more information about this command, see Sketch Circles in this chapter.
- ***Arcs*** Sketches arcs. For more information about this command, see Sketch Arcs in this chapter.
- ***Text*** Adds soil labels, project labels or text labels to the drawing. For more information about this command, see Sketch Text in this chapter.
- ***Axes*** Sketches axes around a section of the drawing. For more information about this command, see Sketch Axes in this chapter.

Sketch Lines

☐ Sketches straight lines.

➢ **To sketch a line on the drawing:**

1. Choose the Lines command from either the Sketch menu or from the Mode toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Sketch Lines" is the current mode.

2. Click at the starting point of the line.

   As you move the cursor, a black line appears, indicating you are sketching a line.

3. Click at the next point of the line.

4. Click at all remaining points on the line if you are not sketching a straight line.

5. Press ESC or click the right mouse button to finish sketching lines.

Comments

If the Snap Grid button in the Grid toolbar is selected, the cursor will snap to a grid point each time you click at a point.

Lines can be moved, resized, or deleted using the Modify Objects command.

Sketch Circles

☐ Sketches circles.

➢ **To sketch a circle on the drawing:**

1. Choose the Circles command from either the Sketch menu or from the Mode toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Sketch Circles" is the current mode.

2. Click at the center point of the circle.

   As you move the cursor, a circle appears, indicating you are defining the radius of the circle.

3. Click at the desired radius of the circle.

   The circle is drawn.

4. Repeat Steps 2 to 3 for as many circles as you wish to sketch.

5. Press ESC or click the right mouse button to finish sketching circles.

Comments

If the Snap Grid button in the Grid toolbar is selected, the cursor will snap to a grid point each time you click at a point.

Circles can be moved, resized, or deleted using the Modify Objects command.

Sketch Arcs

☐ Sketches arcs.

➢ **To sketch an arc on the drawing:**

1. Choose the Arcs command from either the Sketch menu or from the Mode toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Sketch Arcs" is the current mode

2. Click at the center point of the arc.

   As you move the cursor, a circle appears, indicating you are defining the radius and first endpoint of the arc.

3. Click at the first endpoint of the arc.

   A line is drawn from the center of the arc to the first endpoint. As you move the cursor, another line appears, indicating you are defining the second endpoint of the arc.

4. Move the cursor counterclockwise around the circle and click at the second endpoint of the arc.

   An arc is drawn from the first endpoint counterclockwise to the second endpoint.

5. Repeat Steps 2 to 4 for as many arcs as you wish to sketch.

6. Press ESC or click the right mouse button to finish sketching arcs.

Comments

If the Snap Grid button in the Grid toolbar is selected, the cursor will snap to a grid point each time you click at a point.

Arcs can be moved, resized, or deleted using the Modify Objects command.

Sketch Text

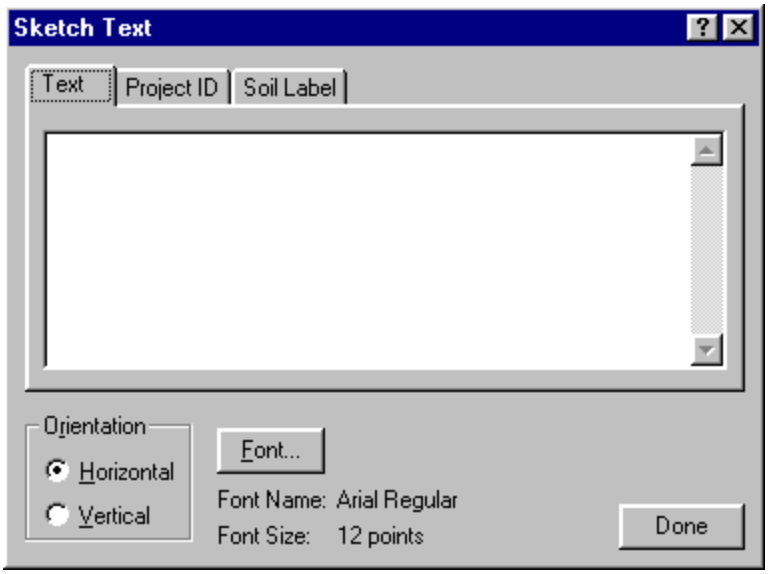
☐ Adds soil labels, project labels, probability labels or text labels to the drawing.

The Sketch Text command can be used to place the following types of text labels on your drawing:

- ***Plain Text Label*** Allows you to type any text and place it on the drawing. You can also import text from other Windows applications via the Windows clipboard and place it on your drawing.
- ***Project ID Label*** Allows you to label the drawing with the current project settings. When you change the project settings using KeyIn Project ID, the corresponding label will be updated automatically with the new project information.
- ***Soil Label*** Allows you to label each soil layer with its soil properties. When you change a soil's properties using KeyIn Soil Properties, the corresponding soil label will be updated automatically with the new properties.
- ***Probability Label*** Allows you to label the drawing in CONTOUR with the computed probabilistic results. When you reanalyze a probabilistic problem using SOLVE, the corresponding Probability label will be updated automatically with the new probabilistic results.

➢ **To place a plain text label on the drawing:**

1. Choose the Text command from either the Sketch menu or from the Mode toolbar. The following dialog box appears:



2. Select the Text tab at the top of the dialog box, if it isn't already selected.

   An edit window is displayed in the dialog box.

3. In the edit window, type the text that you wish to sketch. You can type more than one line of text by pressing the ENTER key after each line.

4. If you wish to sketch text that is in the Windows Clipboard, click the right mouse button in the edit window and select Paste from the pop-up menu; any text in the Windows Clipboard is displayed in the Sketch Text edit window. This feature allows you to place text from another Windows application, such as a word processor, into SLOPE/W.

   You can also copy the text in the edit window to the Windows Clipboard by selecting the text, clicking the right mouse button in the edit window, and selecting Copy from the pop-up menu.

5. Specify the text orientation by selecting Horizontal or Vertical.

6. Move the cursor into the SLOPE/W window and click at the position where you wish the text to appear.

   The text label is placed above and to the right of the selected position, if the label orientation is horizontal; a vertical label is placed above and to the left of the selected position.

   The text font information is displayed in the dialog box underneath the Font button.

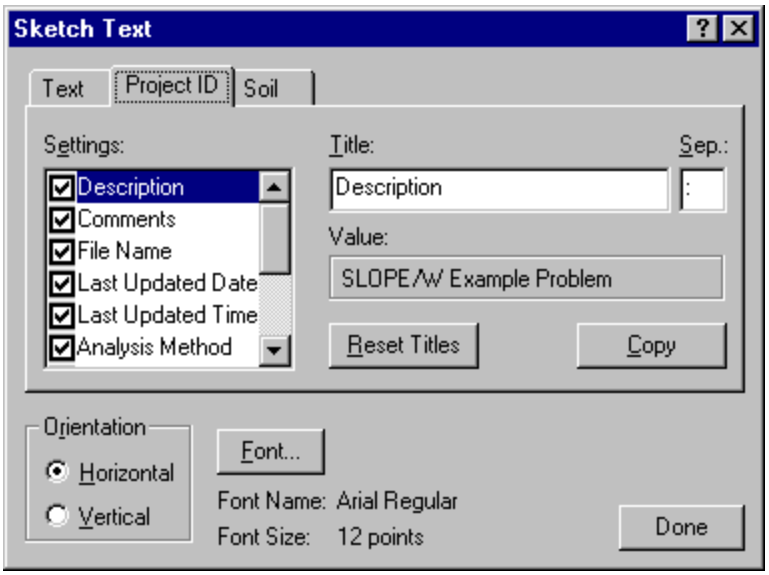
7. Repeat Step 6 if you wish to place the text at another position on the drawing.

8. To finish placing text, press ESC or select another operating mode from the Mode toolbar.

➢ **To place a Project ID label on the drawing:**

1. Choose the Text command from either the Sketch menu or from the Mode toolbar. The Sketch Text dialog box appears.

2. Select the Project ID tab at the top of the dialog box, if it isn't already selected. The Project ID information is displayed in the dialog box as follows:



3. In the Settings list box, check the box next to each parameter that you wish to include in the Project ID label.

4. To change the title for a parameter, select the parameter in the Settings list box and then type a new title in the Title edit box. You can display a parameter without any title by removing the text from the Title edit box.

5. To change the separator between a parameter and its title, type a new character (or several characters) in the Sep. edit box. The new separator will be used for each parameter in the Project ID label.

6. To reset all parameter titles to the default titles, select the Reset Titles button.

7. To copy the current Project ID label to the Windows clipboard, select the Copy button. You can then paste the Project ID label into other Windows applications.

8. Specify the text orientation by selecting Horizontal or Vertical.

9. Move the cursor into the SLOPE/W window and click at the position where you wish the Project ID label to appear.

   The Project ID label is placed above and to the right of the selected position, if the label orientation is horizontal; a vertical label is placed above and to the left of the selected position.

   The text font information is displayed in the dialog box underneath the Font button.

10. Repeat Step 9 if you wish to place the Project ID label at another position on the drawing.

11. To finish placing text, press ESC or select another operating mode from the Mode toolbar.

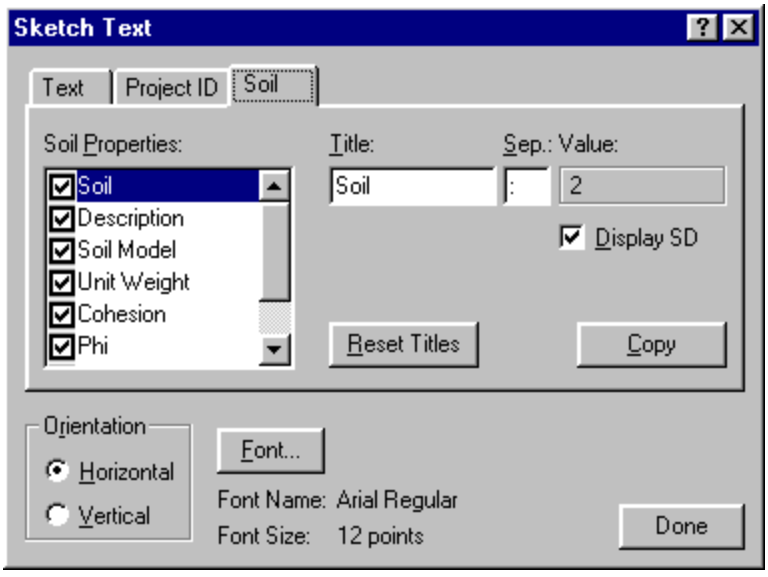
NOTE: If you change your project settings, your Project ID label will be automatically updated to show the current project settings. You can use KeyIn Project ID, KeyIn Analysis Control, KeyIn Analysis Method, KeyIn Load: Seismic Load, or File Save As to change the project settings.

➢ **To place a soil label on the drawing:**

1. Choose the Text command from either the Sketch menu or from the Mode toolbar. The Sketch Text dialog box appears.

2. Select the Soil tab at the top of the dialog box, if it isn't already selected. An empty soil information property sheet is displayed in the dialog box.

3. In the SLOPE/W window, move the cursor inside a soil layer or on top of a soil line. (Notice that the cursor changes to a black selection arrow when it is inside a soil layer.) Click the left mouse button to select the soil. The soil is shaded with a diagonal hatch pattern, and the soil line and points are highlighted. The soil properties are displayed in the Sketch Text dialog box as follows:



4. In the Soil Properties list box, check the box next to each parameter that you wish to include in the Soil label.

5. Check the Display SD check box if you wish to display the standard deviations with each soil property. This option is only available is you have selected a probability analysis with KeyIn Analysis Control.

6. To change the title for a parameter, select the parameter in the Soil Properties list box and then type a new title in the Title edit box. You can display a parameter without any title by removing the text from the Title edit box.

7. To change the separator between a parameter and its title, type a new character (or several characters) in the Sep. edit box. The new separator will be used for each parameter in the Soil label.

8. To reset all parameter titles to the default titles, select the Reset Titles button.

9. To copy the current Soil label to the Windows clipboard, select the Copy button. You can then paste the Soil label into other Windows applications.

10. Specify the text orientation by selecting Horizontal or Vertical.

11. Move the cursor inside the selected soil layer or outside the soil geometry, so that the cursor is shown as a cross-hair. Then, click the left mouse button to place the soil label.

    NOTE: When you move the cursor inside a soil layer that isn't already selected, the cursor changes to a black selection arrow. This indicates that a label will not be placed if you click the left mouse button; instead, a new soil will be selected.

    The Soil label is placed above and to the right of the selected position, if the label orientation is horizontal; a vertical label is placed above and to the left of the selected position.

    The text font information is displayed in the dialog box underneath the Font button.

12. Repeat Step 11 if you wish to place the Soil label at another position on the drawing.

13. Repeat Steps 3 to 12 if you wish to place a label on another soil.

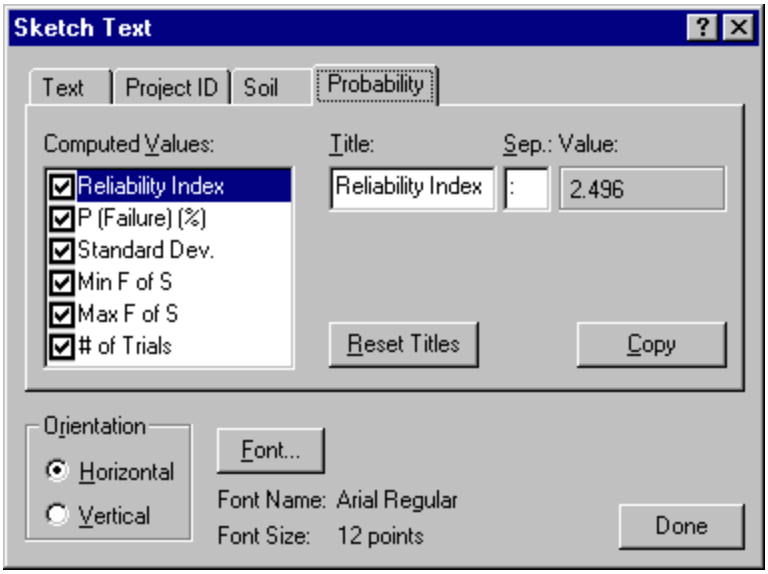
14. To finish placing text, press ESC or select another operating mode from the Mode toolbar.

NOTE: If you change the soil properties using KeyIn Soil Properties, each soil label will be automatically updated to show the new properties.

- **To place a Probability label on the drawing (in CONTOUR only):**

1. Choose the Text command from either the CONTOUR Sketch menu or from the CONTOUR Mode toolbar. The Sketch Text dialog box appears.

2. Select the Probability tab at the top of the dialog box. The computed Probability information is displayed in the dialog box as follows:



NOTE: The Probability tab is only displayed in the CONTOUR Sketch Text dialog box if a probability analysis was selected in DEFINE using KeyIn Analysis Control.

3. In the Computed Values list box, check the box next to each parameter that you wish to include in the Probability label.

4. To change the title for a parameter, select the parameter in the Computed Values list box and then type a new title in the Title edit box. You can display a parameter without any title by removing the text from the Title edit box.

5. To change the separator between a parameter and its title, type a new character (or several characters) in the Sep. edit box. The new separator will be used for each parameter in the Probability label.

6. To reset all parameter titles to the default titles, select the Reset Titles button.

7. To copy the current Probability label to the Windows clipboard, select the Copy button. You can then paste the Probability label into other Windows applications.

8. Specify the text orientation by selecting Horizontal or Vertical.

9. Move the cursor into the CONTOUR window and click at the position where you wish the Probability label to appear.

   The Probability label is placed above and to the right of the selected position, if the label orientation is horizontal; a vertical label is placed above and to the left of the selected position.

   The text font information is displayed in the dialog box underneath the Font button.

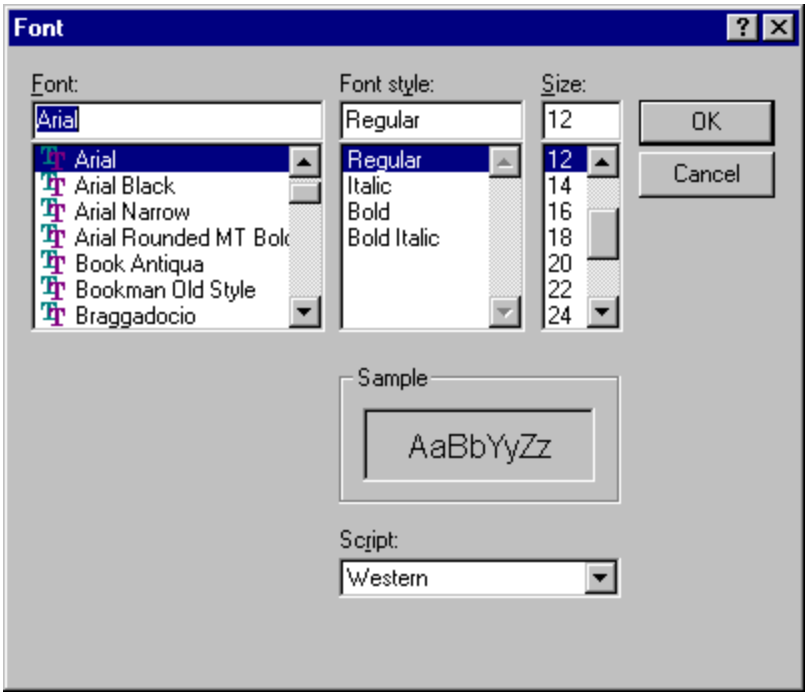
10. Repeat Step 9 if you wish to place the Probability label at another position on the drawing.

11. To finish placing text, press ESC or select another operating mode from the Mode toolbar.

NOTE: If you reanalyze your problem using SOLVE, your Probability label will be automatically updated with the new probabilistic results.

➢ **To change the font of the text label:**

1. In the Sketch Text dialog box, click on the Font button to change the text font. The following dialog box appears:



All the fonts that are currently installed in Windows are displayed in the Font list box. To install or delete fonts, you must use the Windows Control Panel. See the Windows online help for more information on Control Panel.

2. Select the desired font in the Font list box and style in the Font Style list box.

3. Select a point size from the Size list box or type any point size in the Size edit box.

   Points are the units commonly used for font size (72 points is equal to 1 inch). The point size that you enter represents the height of the text at a zoom factor of 1.0.

4 Select OK to return to the Sketch Text dialog box. The name and size of the selected font is displayed underneath the Font button.

5. Move the cursor into the SLOPE/W window and click at the position where you wish the text label to appear.

   The text label is placed on the drawing using the selected font.

Comments

Text labels can be moved, resized, or deleted using the Modify Objects command.

Text labels can be changed using the Modify Text command. If you modify a soil label, for example, you can add or remove any of the soil properties that are displayed on the label.

If the Snap Grid button in the Grid toolbar is selected, the cursor will snap to a grid point each time you click at a point.

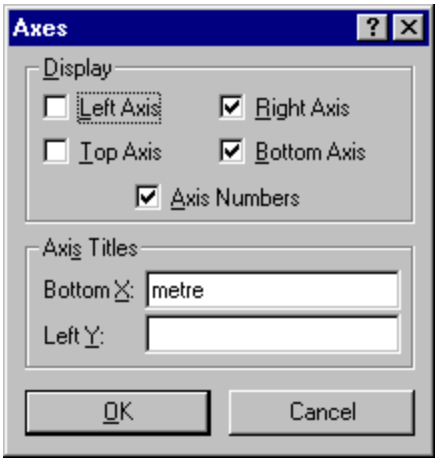
Sketch Axes

☐ Sketches axes surrounding a section of the drawing.

Sketching an axis on the drawing facilitates viewing the drawing and interpreting the drawing after it is printed.

➢ **To sketch axes:**

1. Choose the Axes command from either the Sketch menu or from the Mode toolbar. The following dialog box appears:



2. In the Display group box, check axes that you wish to sketch. If all four sides are selected, the axes will form a box.

3. Check the Axis Numbers check box if you desire each tick mark on the axis to be labelled with its value.

4. Type an appropriate title for the bottom X-axis in the Bottom X edit box, if desired.

5. Type an appropriate title for the left Y-axis in the Left Y edit box, if desired.

6. Select OK. The cursor changes from an arrow to a cross-hair and the status bar indicates that "Sketch Axes" is the current mode.

7. To define the rectangular region over which to sketch the axes, hold the left mouse button down at the top-left corner of the axes region, but do not release it. As you move the mouse, a rectangle appears.

8. Drag the mouse to the bottom-right corner of the axes region and release the left mouse button.

   Axes are generated within the region.

Comments

The number of increments along each axis is calculated by SLOPE/W when the axes are generated. Choose the Set Axes command if you wish to override these values.

If the Snap Grid button in the Grid toolbar is selected, the cursor will snap to a grid point each time you click at a point. This is useful for sketching an axis with exact increments.

Axes can be moved, resized, or deleted using the Modify Objects command.

The View Preferences command allows you to change the font and the size of the axes numbers and labels.

The Modify Menu

Use the Modify menu to move, resize, or delete any group of selected objects or to change text items on the drawing.

- ***Objects*** Moves, resizes, or deletes any group of selected objects, such as points, soil lines, or sketch objects. For more information about this command, see Modify Objects in this chapter.
- ***Text*** Changes text labels that were placed on the drawing using the Sketch Text command. For more information about this command, see Modify Text in this chapter.
- ***Pictures*** Changes the ordering, file name, or scale of any picture imported with the File Import: Picture command. For more information about this command, see Modify Pictures in this chapter.

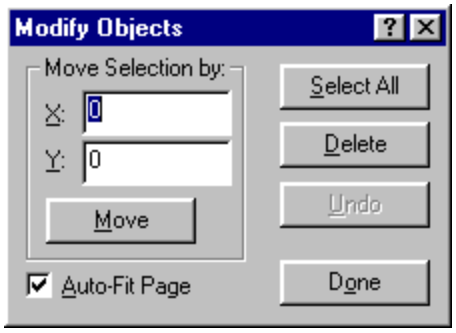
Modify Objects

☐ Moves, re-sizes, or deletes any group of selected objects, such as points, soil lines, or sketch objects.

Modify Objects is a powerful command that allows you to select objects on the drawing for moving, resizing, or deletion. It provides an interactive method of changing the engineering coordinates of any object or group of objects. Objects are defined as any item displayed on the drawing at specified engineering coordinates.

Most objects in SLOPE/W, such as soil lines, are referenced to points. When you use Modify Objects to select a soil line, for example, SLOPE/W will select all the points that define the soil line. When you drag the line to a new position, SLOPE/W moves each point on the line to its new position. Other object types used in DEFINE are text, lines, circles, arcs, and the scaled axes. In CONTOUR, you can only modify Sketch objects and the axes.

When you choose Modify Objects, the following dialog box appears:



Move Selection by X The *x*-distance, in engineering coordinates, to move the selected objects.

Move Selection by Y The *y*-distance, in engineering coordinates, to move the selected objects.

Move When this button is pressed, the selected objects are moved by the distance specified in the X and Y edit boxes.

Auto-Fit Page When this option is checked and any objects are moved or scaled, the working area page size changes, if necessary, to encompass any objects that lie outside of the working area. If all objects are moved outside the working area, then the working area moves with the objects but doesn't change in size.

Select All When this button is pressed, all objects currently displayed on the drawing are selected. If you wish to select all objects of a specific type, such as soil lines, then use the View Preferences toolbar to only view soil lines and then press Select All.

Delete When this button is pressed, all selected objects are deleted from the problem.

Undo When this button is pressed, the last action is reversed and all objects are restored to their previous state. The Undo button is then disabled, since it cannot undo any actions before the last one.

Done When this button is pressed, you are exited from the Modify Objects operating mode. Alternatively, you can press the ESC key or select another operating mode from the Mode toolbar.

➢ **To modify objects:**

1. Choose the Objects command from either the Modify menu or from the Mode toolbar.

   The cursor changes from a white arrow to a black arrow, the status bar indicates that "Modify Objects" is the current operating mode, and the Modify Objects dialog box appears.

2. In the SLOPE/W window, select the objects to modify using the left mouse button.

3. Apply the desired action to the selected objects, such as moving, scaling, or deleting them. For example, to delete the selected objects, select Delete in the dialog box or press the DELETE key on the keyboard.

4. To undo the last action, select the Undo button in the dialog box or press CTRL-Z on the keyboard. For example, if you deleted a group of objects and then select Undo, the objects will reappear.

5. If necessary, repeat Steps 2 to 4 for all objects that you wish to modify.

6. Select Done or press the ESC key to finish modifying objects.

NOTE: You can only undo one level of modification operations. For example, if you move an object and then delete it, pressing Undo will recreate the object, and the Undo button will be disabled. You cannot restore the object to the position it was at before it was moved.

Selecting Objects

➢ **To select objects:**

- Click on any object with the left mouse button; the object is selected.

  -- or --

- Hold down the left mouse button and drag a rectangle around a group of objects; all objects completely inside the rectangle are selected.

  -- or --

- Click on the Select All button in the dialog box; all objects currently displayed on the drawing are selected.

  -- or --

- Select a series of points along a straight line by holding down the SHIFT key and clicking on the first and last points in the line; all points that lie along the line are selected.

Each time a new selection is made, all other objects are unselected. If you wish to keep the previous object selection, hold down the CTRL key while you select more objects.

Selected objects are highlighted with a graphic symbol, usually a hollow rectangle; selected points are displayed as large rectangles. Handles are drawn around the boundary of all selected objects - at the corners and at each side in between. These handles are used to resize and reshape the selected objects.

TIP: When several objects are displayed on top of each other, it can be difficult to select the desired object. Use the View Preferences Toolbar to hide or show only the object types that you wish to modify. For example, before moving a sketch line, uncheck the View Points and View Lines toolbar buttons to hide the points and lines; this will prevent you from inadvertently selecting points or soil lines when you are trying to select sketch lines.

Moving Objects

➢ **To move objects:**

- Click on any unselected object, holding down the left mouse button, and drag the object to its new position. A dashed, rectangular border appears around the selected object and moves as you drag the object.

  -- or --

- Click down on an object that is already selected and drag the selected objects to their new positions. A dashed, rectangular border appears around the group of selected objects and moves as you drag the objects.

  -- or --

- In the Modify Objects dialog box, type in the *x*- and *y*-distance (in engineering coordinates) to move all selected objects and press the Move button.

If the background grid is turned on, the selected object being dragged by the mouse will be snapped to the closest grid point when the left mouse button is released. For objects such as sketch lines, the corner of the object that is nearest to the mouse cursor is snapped to the closest grid point; for text items, the bottom-left corner of the text is snapped to the grid point; for circles and arcs, the center is snapped to the grid point.

➢ **To move the entire drawing a specified distance:**

1. Make sure all object types are currently displayed in the View Preferences toolbar or dialog box.
2. In the Modify Objects dialog box, press the Select All button. All objects on the drawing are selected.
3. In the Move Selection By edit boxes, type the *x*- and *y*-distance, in engineering coordinates, to move the drawing. For example, if you defined your problem at an origin of (0,0) and wish to change the elevation at the origin to 400 meters, type **400** in the Y edit box.
4. Make sure the Auto-Fit Page option is checked in the dialog box.
5. Press the Move button. All objects in the drawing are moved by the specified distance, and the working area is adjusted as necessary to fit around all objects in the drawing.

Resizing Objects

➢ **To resize objects:**

1. Select the objects to resize.
2. Click down on one of the eight handles displayed around the selected objects.

   The cursor changes to an arrow, indicating the direction in which the objects will be scaled. A dashed, rectangular boundary is displayed around the selected objects.
3. Drag the mouse in the desired direction. As you drag, the rectangular boundary is resized.
4. Release the left mouse button when you are satisfied with the new scale.

   All the selected objects are modified to fit inside the new rectangular boundary.

5. If you wish to return the selected objects to their previous size, press CTRL-Z or press the Undo button in the Modify Objects dialog box.

Deleting Objects

➢ **To delete objects:**

1. Select the objects to delete.
2. To delete the objects, press the DELETE key or press the Delete button in the dialog box.

   All the selected objects are deleted from the problem.
3. If you wish to recreate the deleted objects, press CTRL-Z or press the Undo button in the dialog box.

NOTE: When you delete objects that are defined by points, such as soil lines, SLOPE/W deletes the points used to define the object. Any other objects that are defined by these points will also be deleted. For example, if you delete a soil line that has a line load attached to one of its points, the line load will also be deleted.

Modify Text

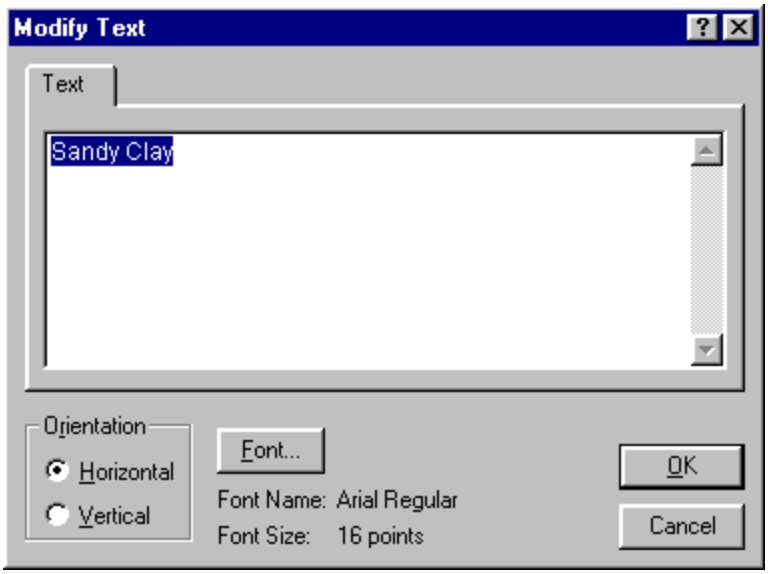
☐ Changes text labels that were placed on the drawing using the Sketch Text command.

➢ **To modify text:**

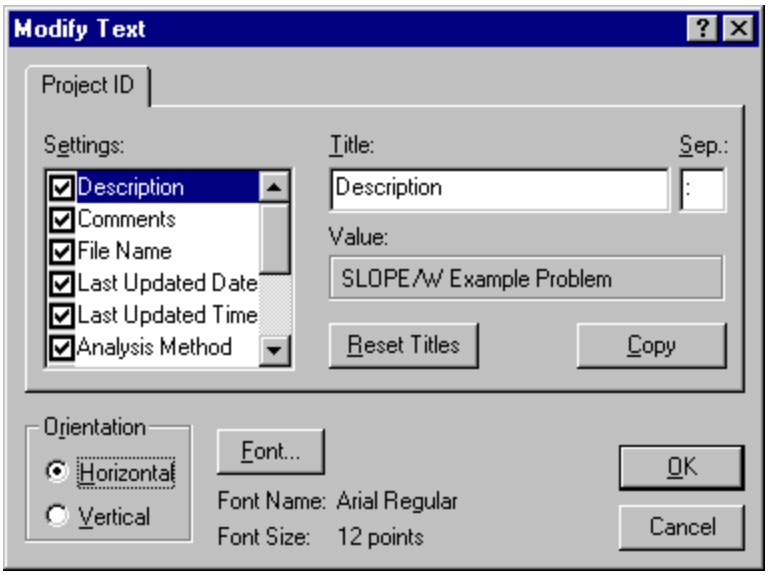
1. Choose the Text command from either the Modify menu or from the Mode toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Modify Text" is the current operating mode.
2. Click the left mouse button inside a text label on the drawing that you wish to modify.

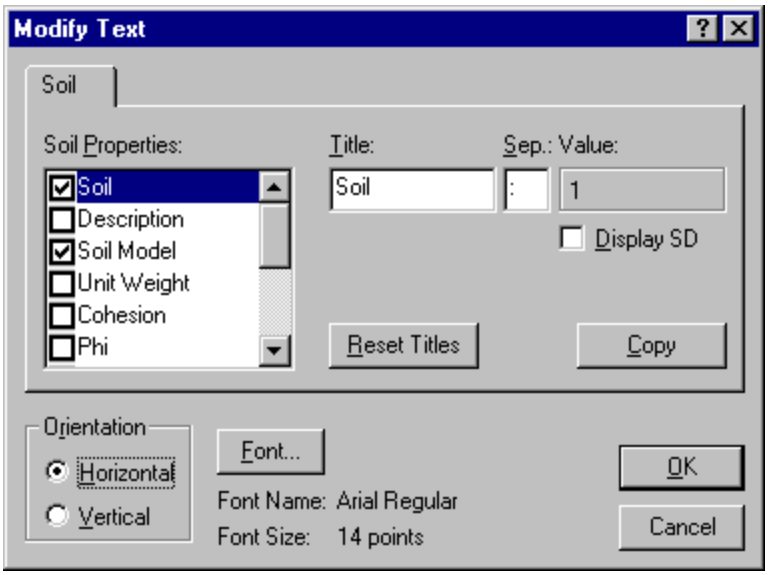
   If you clicked on a plain text label, the following dialog box appears:



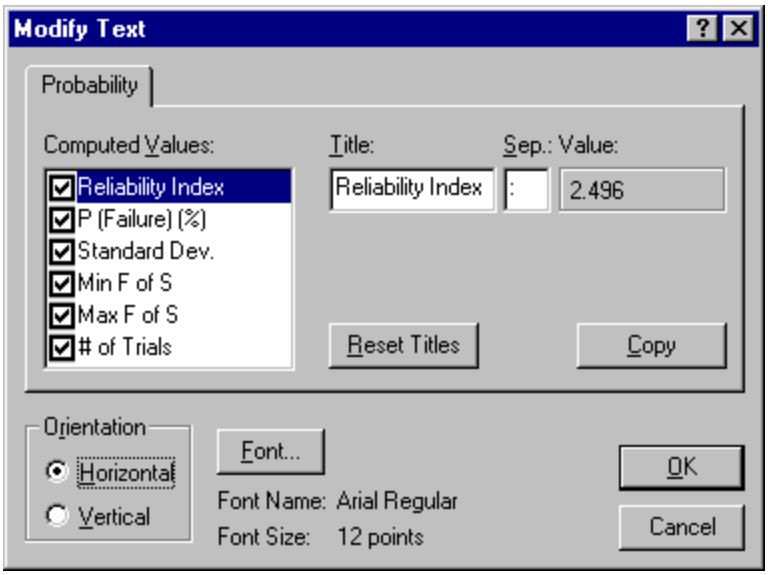
If you clicked on a Project ID label, the following dialog box appears:



If you clicked on a plain text label, the following dialog box appears:



If you clicked on a Probability label in CONTOUR, the following dialog box appears:



3. Change any of the text label information. For information on how to change each type of text label, see the Sketch Text section.

4. Select OK when you are finished changing the text label information. The text is redrawn to reflect the changes made.

5. Repeat Steps 2 to 4 for each text label to modify.

6. To finish modifying text, press the ESC key or select another operating mode from the Mode toolbar.

Modify Pictures

☐ Changes the ordering, file name, or scale of any picture imported with the File Import: Picture command.

The Modify Pictures command allows you to change the following attributes of imported pictures:

- The order in which pictures are displayed on the drawing can be changed. This is useful if a picture overlaps with another picture or with part of the drawing.

- The file name that a picture is linked to can be changed. This is useful if you wish to rename or move the linked file or if you have an updated file that you wish to link the picture to.

- The scale (i.e., the size) of a picture can be changed by mapping engineering coordinates on the picture to coordinates on the drawing. This is useful if you have imported a picture of your slope stability problem and you wish to define your SLOPE/W geometry on top of the imported picture.

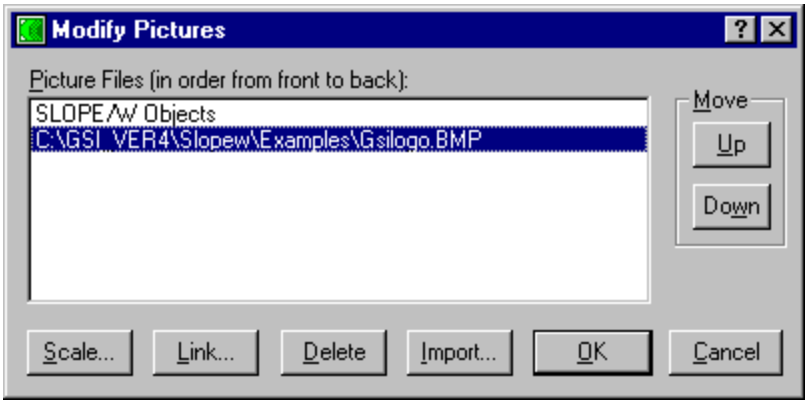
If you wish to move a picture or change its size, choose the Modify Objects command.

The Modify Pictures command is disabled if no pictures were previously imported with the File Import: Picture command.

➢ **To select a picture to modify:**

1. Choose the Pictures command from either the Modify menu or from the Mode toolbar.

   The cursor changes to a black selection arrow and the status bar indicates that "Modify Pictures" is the current operating mode. The following dialog box appears:



   The Picture Files list box displays a list of the imported pictures. The SLOPE/W Objects item is displayed so that you can move a picture in front of or behind the rest of the SLOPE/W drawing.

2. To see all the picture file information, re-size the dialog box by dragging one of the window edges until all the information is displayed in the Picture Files list box.

3. Select the picture that you wish to modify. You can either select the picture file name in the dialog box or you can click on the picture itself in the SLOPE/W window.

   A rectangle is drawn around the selected picture in the SLOPE/W window.

➢ **To change the order in which the selected picture is displayed on the drawing:**

1. If the selected picture is obscured by other objects on the drawing, select the Up button to display it on top of other pictures in the drawing.

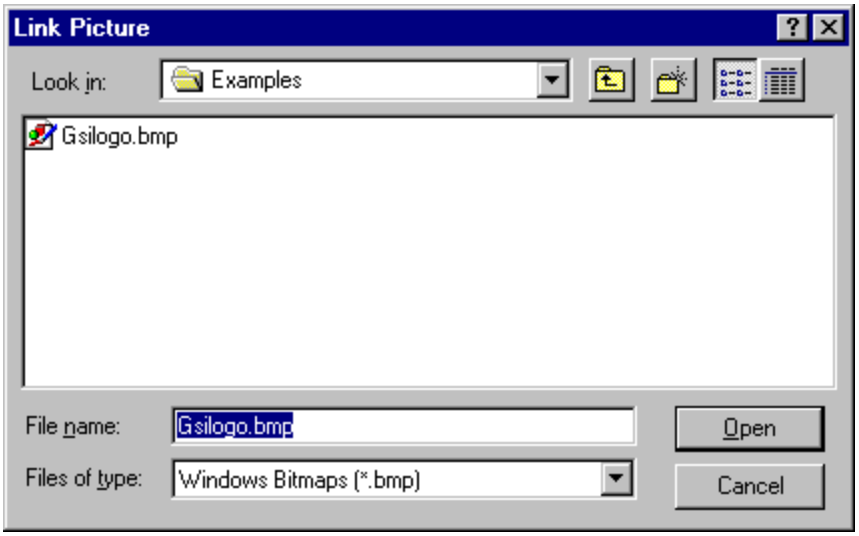
   Each time you move the picture up in the list, it is redrawn in the SLOPE/W window. You can continue selecting Up until the picture is displayed on top of all other objects, including the SLOPE/W drawing itself.

2. Select Down if you wish to move the picture towards the back of the drawing.

➢ **To change the file name that the selected picture is linked to:**

1. Once you have selected a picture, click on the Link button in the Modify Pictures dialog box.

The following dialog box appears:



NOTE: The SLOPE/W Link Picture dialog box is a common dialog used by many other Windows applications. To get help on using the dialog box, click on the question-mark in the top-right corner; your cursor then becomes a question mark. Then, click on the dialog control that you need explained; a pop-up window appears with a description of the dialog control. Click anywhere else in the dialog box to remove the pop-up window.

2. Select the new file name that you wish to link to the picture. In the Files of Type drop-down list box, select the format of the picture files you wish to display.

3. Once you have specify the new picture file name in the File Name edit box, select Open.

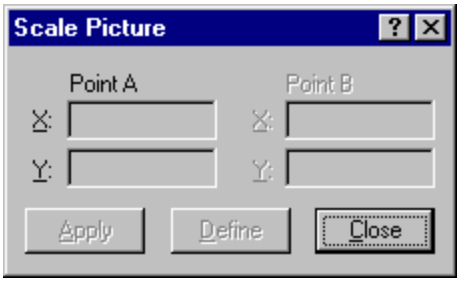
   The new file name for the selected picture is displayed in the Modify Pictures dialog box. The new picture is shown on the SLOPE/W drawing.

➢ **To scale the selected picture to match the current engineering scale:**

Scaling a picture is useful when you have imported a picture of your slope stability problem and you wish to define your SLOPE/W geometry on top of the imported picture.

1. Once you have selected a picture, click on the Scale button in the Modify Pictures dialog box.

   The following dialog box appears, and the cursor changes to a cross-hair, indicating that you need to locate two reference points on the picture:



2. Click on the selected picture to define Point A, the first reference point.

Once you have defined Point A, its SLOPE/W engineering coordinates are displayed in the Point A edit boxes. The reference point is marked on the drawing with the letter A and a cross-hair.

NOTE: Typically, Point A should be located near the lower-left corner of the picture at a position where you know the engineering coordinates on the picture. For example, if you have imported a picture of a slope stability problem that was created from an origin of (0,0), define Point A at the (0,0) position on the picture. (The actual SLOPE/W engineering coordinates of Point A, as displayed in the Modify Pictures dialog box, will be different; for example, they may be shown in the edit boxes as (2,2).)

3. Click on the selected picture to define Point B, the second reference point.

   The engineering coordinates of the second reference point are displayed in the Point B edit boxes. The second reference point is marked on the drawing with the letter B and a cross-hair.

   Once you have defined Point B, its SLOPE/W engineering coordinates are displayed in the Point B edit boxes. The reference point is marked on the drawing with the letter B and a cross-hair.

   NOTE: Typically, Point B should be located near the upper-right corner of the picture at a position where you know the engineering coordinates on the picture. For example, if you have imported a picture of a slope stability problem that extends to an elevation of 15 and a width of 20, define Point B at the (15,20) position on the picture. (The actual SLOPE/W engineering coordinates of Point B, as displayed in the Modify Pictures dialog box, will be different; for example, they may be shown in the edit boxes as (30,40).)

4. Type the new coordinates for Point A in the corresponding X and Y edit boxes.

   For example, if you defined Point A at the (0,0) coordinate on the picture, enter (0,0) as the new coordinates for Point A.

5. Type the new coordinates for Point B in the corresponding X and Y edit boxes.

   For example, if you defined Point B at the (15,20) coordinate on the picture, enter (15,20) as the new coordinates for Point A.

6. Select the Apply button to resize or move the picture.

   SLOPE/W matches Point A and B on the picture to their new coordinates. For example, assume that Point A on the picture is located at (2,2) on the drawing and Point B is located at (30,40). You have just entered (0,0) as the new coordinates for Point A and (15,20) as the new coordinates for Point B. When you click on Apply, the picture is moved and resized so that Point A on the picture is now located at (0,0) and Point B is now located at (15,20). You can verify that this is true by moving your cursor above Point A on the picture and checking that the SLOPE/W status bar displays an X and Y position of (0,0).

7. If the picture was not scaled properly, repeat Steps 4 to 6 in order to enter new coordinates for Point A and B. If you need to reposition the picture reference points, repeat Steps 2 to 6.

8. When you are satisfied with the scaled picture, select the Close button in order to return to the Modify Pictures dialog box.

➢ **To delete a selected picture from the drawing:**

- Select the Delete button in the Modify Pictures dialog box. The picture will be removed from the drawing and from the Picture Files list box.

➢ **To import a new picture into the drawing:**

- Select the Import button in the Modify Pictures dialog box. This button is a shortcut for the File Import: Picture command. See this command for more information on importing a picture into the drawing.

  Once you have placed the imported picture into the drawing, the picture file name will be displayed in the Modify Pictures dialog box.

NOTE: Once you are finished modifying pictures, be sure to press the OK button in the dialog box to save your changes. All changes made to the pictures will be lost if you select Cancel, press the ESC key, or select another operating mode from the Mode toolbar.

The Tools Menu

Use the Tools menu to perform tasks such as verifying the problem data and switching to SOLVE or CONTOUR.

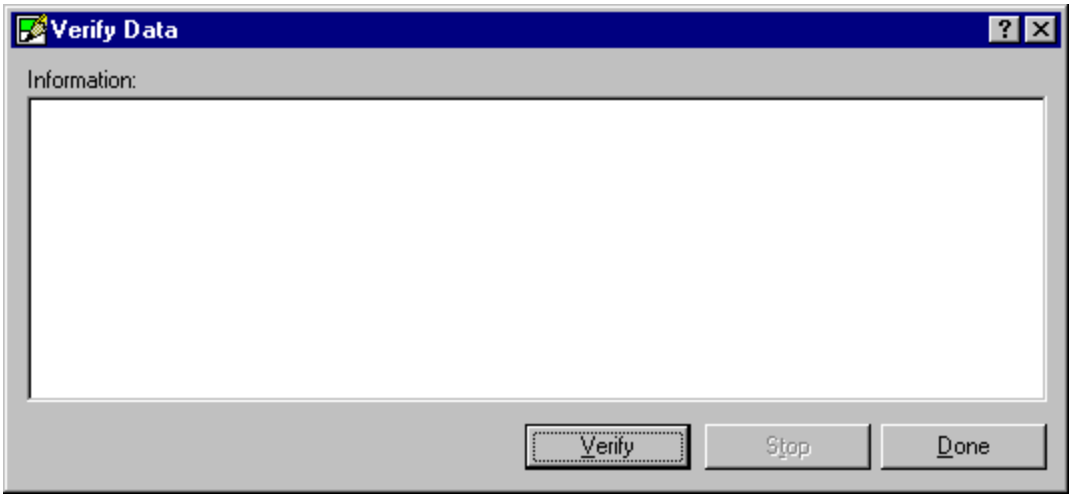
- ***Verify*** Verifies the correctness of the geometric data. For more information about this command, see Tools Verify in this chapter.
- ***SOLVE*** Launches SOLVE and opens the file currently being edited in DEFINE. For more information about this command, see Tools SOLVE in this chapter.
- ***CONTOUR*** Launches CONTOUR and opens the file currently being edited in DEFINE. For more information about this command, see Tools CONTOUR in this chapter.

Tools Verify

☐ Verifies the correctness of the geometric data.

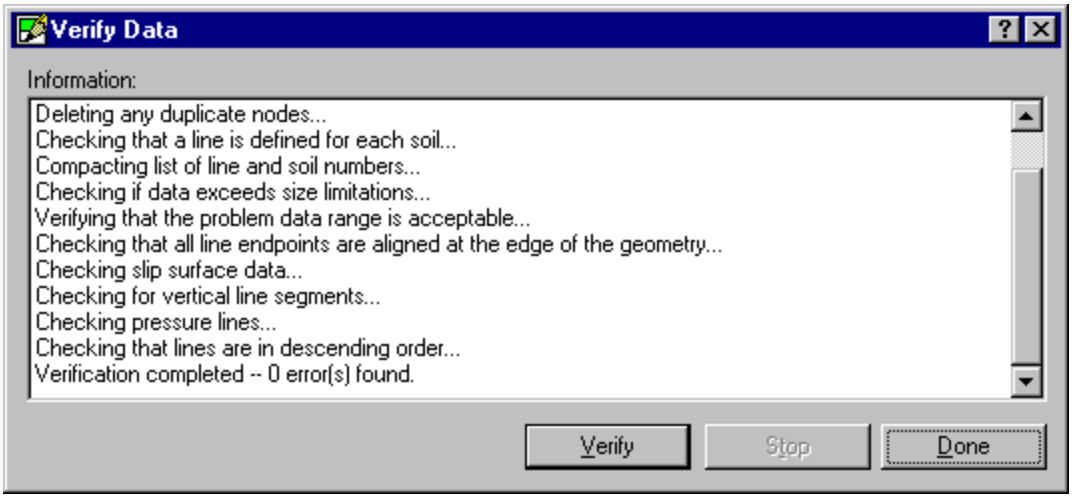
➢ **To verify the data:**

1. Choose the Verify command from either the Tools menu or from the Standard toolbar. The following dialog box appears:



2. Select the Verify button to verify the geometric data definition.

   Messages appear in the Information list box stating which verification step is being performed. Error messages will also appear in the list box as necessary. Select Stop if you wish to stop the verification. A beep is sounded when the verification is finished:


Verify Data
Information:
Deleting any duplicate nodes...
Checking that a line is defined for each soil...
Compacting list of line and soil numbers...
Checking if data exceeds size limitations...
Verifying that the problem data range is acceptable...
Checking that all line endpoints are aligned at the edge of the geometry...
Checking slip surface data...
Checking for vertical line segments...
Checking pressure lines...
Checking that lines are in descending order...
Verification completed -- 0 error(s) found.
Verify
Stop
Done

For each error found in the data, an error message is displayed in the dialog box. The total number of errors found is displayed as the last line in the dialog box.

3. To see all the verification messages in the list box, re-size the dialog box by dragging the bottom edge of the window down until all information is displayed.

4. Select Done when you are finished viewing the messages in the Verify Data dialog box.

SLOPE/W performs the following steps when verifying the data:

1. Checks that data does not exceed size limitations. SLOPE/W's data size limitations are given below in Table 4.2

2. Searches for points with the same coordinates and replaces the duplicate points with one point number, modifying all data accordingly.

3. Compresses line and soil numbers into a continuous sequence.

4. Checks that a line is defined for every soil layer.

5. Checks that the endpoints of all lines, piezometric lines, pressure contour lines, surface pressure lines, and the tension crack line begin at the same *x*-coordinate and end at the same *x*-coordinate.

6. Checks that all lines, piezometric lines, pressure contours, fully specified slip surfaces, surface pressure lines, and tension crack line are defined from left to right, and that no vertical line segments are contained in any of these lines. The *x*-coordinate of each point in a line must be greater than the *x*-coordinate of the previous point in the line.

7. Checks that the geometry lines are specified in descending order, starting with the ground or water surface and ending with the bottom soil layer.

8. Checks that radius lines and a grid of slip surface centers are defined if the slip surface option selected is Grid and Radius.

9. Checks that an axis point and at least one specified slip surface are defined if the slip surface option selected is Fully Specified. It also check that each fully specified slip surface begins and ends above the top soil line.

10. Checks that an axis point and a left and right block of slip surface intersection points are defined if the slip surface option selected is Block Specified.

11. Checks the range of the tension crack angle and the slip surface projection angles, if they are defined.

    If the angle is defined on the **left** side of the slip surface on the **active** side of the problem (i.e., the slip surface movement is from left to right), the angle must be between 100º and 135º.

    If the angle is defined on the **left** side of the slip surface on the **passive** side, the angle must be between 120º and 180º.

    If the angle is defined on the **right** side of the slip surface on the **active** side of the problem (i.e., the slip surface movement is from right to left), the angle must be between 45º and 80º.

    If the angle is defined on the **right** side of the slip surface on the **passive** side, the angle must be between 0º and 60º.

12. Checks that no surface pressure lines overlap and that all pressure lines lie above the top soil layer (Soil Line 1).

13. Checks that the tension crack line lies below the top soil layer (Soil Line 1) and below any soil layers defined using the No Strength soil model. That is, the tension crack line cannot lie within water.

Table 4.2 SLOPE/W Data Size Limitations

Parameter	Maximum
Points	1000
Lines	50
Soils	50
Pressure Contours	50
Piezometric Lines	50
Fully Specified Slip Surfaces	no limit
Points in a geometry, piezometric, or contour line or a fully specified slip surface	50
Pore-water points (heads, pressures, or r_u coefficients)	50
Line and anchor loads	no limit
Pressure lines	no limit
Slip surface centers in a grid	no limit
Radii per slip surface center	no limit
Slices into which the slip surface can be subdivided	150

NOTE: Verify is a tool to help you with your mesh generation and problem definition. It is a very powerful and useful tool but it does not guarantee that you have an error free mesh or a perfect problem definition. In the end it is still up to you to ensure that the model is correct. Do not make the assumption that everything is perfect after you have run Verify.

Tools SOLVE

☐ Launches SOLVE and opens the file currently being edited in DEFINE.

Use the Tools SOLVE command to quickly start a *new* copy of SOLVE. When you choose Tools SOLVE, you are prompted to save the data file currently being edited in DEFINE; SOLVE will then start and will open this data file.

NOTE: If you are running DEFINE, SOLVE and CONTOUR at the same time and have the same problem opened in all three programs, you do not need to launch SOLVE each time you save your problem in DEFINE; SOLVE will read the new problem data files each time you press the Start button to begin the problem analysis.

In Windows 95 and Windows NT 4.0, use the Task Bar to switch between DEFINE, SOLVE, and CONTOUR.

Tools CONTOUR

☐ Launches CONTOUR and opens the file currently being edited in DEFINE.

Use the Tools CONTOUR command to quickly start a *new* copy of CONTOUR. When you choose Tools CONTOUR, you are prompted to save the data file currently being edited in DEFINE; CONTOUR will then start and will open this data file.

NOTE: If you are running DEFINE, SOLVE and CONTOUR at the same time and have the same problem opened in all three programs, you do not need to launch CONTOUR each time you have re-analyzed the problem with SOLVE. When SOLVE is finished the problem analysis and you select the CONTOUR window, you are prompted to allow CONTOUR to read the newly-computed output files. This will change the contours, vectors, and other results displayed by CONTOUR.

In Windows 95 and Windows NT 4.0, use the Task Bar to switch between DEFINE, SOLVE, and CONTOUR.

The Help Menu

The Help menu commands are:

- ***Help Topics*** Displays on-line help. Use the Help Topics command to access the on-line help system. Help topics may be accessed from the table of contents, from an index, or by searching for specific words. For more information on using Windows help, see the Windows documentation.

- ***Using Help*** Displays the help system with information about using the on-line help system. For more information about using on-line help, see Using On-Line Help in Chapter 2.

- ***About SLOPE/W*** Displays information about SLOPE/W, such as the version and serial number. Use the System Information button in the About dialog box to quickly display information about your computer, such as the version of Windows, the processor type, and the amount of memory.

Chapter 5 SOLVE Reference

Introduction **2**

The File Menu **3**
File Open Data File 3

The Help Menu **6**

Running SOLVE **7**

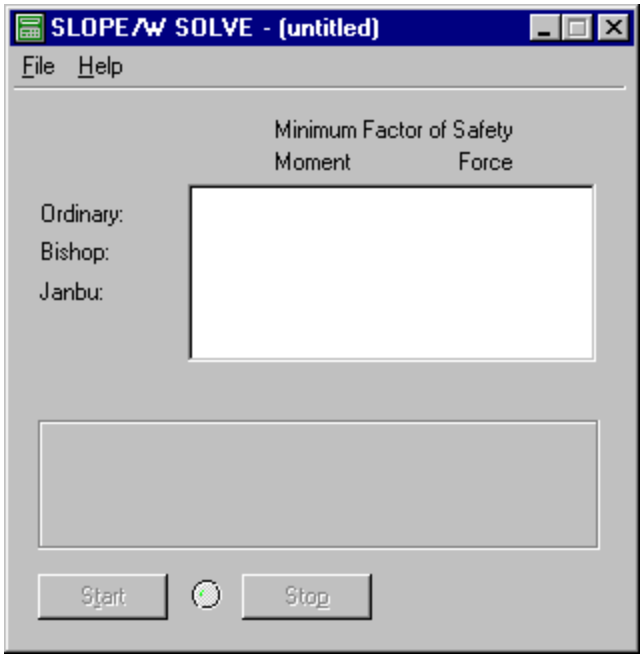
Files Created for Limit Equilibrium Methods **10**
Factor of Safety File - Limit Equilibrium Method 10
Slice Forces File - Limit Equilibrium Method 12
Probability File - Limit Equilibrium Method 15

Files Created for the Finite Element Method **17**
Factor of Safety File - Finite Element Method 17
Slice Forces File - Finite Element Method 17
Probability File - Finite Element Method 19

Introduction

SOLVE is the function that computes the factors of safety, the probabilistic data and the slice data for the critical slip surface after the problem has been defined with the DEFINE function. SOLVE reads the data file created by DEFINE and stores the results in several output files.

The SOLVE window does not display a drawing of the problem, like other SLOPE/W functions, but instead displays a dialog box that shows the minimum factors of safety and the problem file names.



This chapter describes the following:

- How to run SOLVE.
- The information displayed during the processing.
- The output files created by SOLVE.

The File Menu

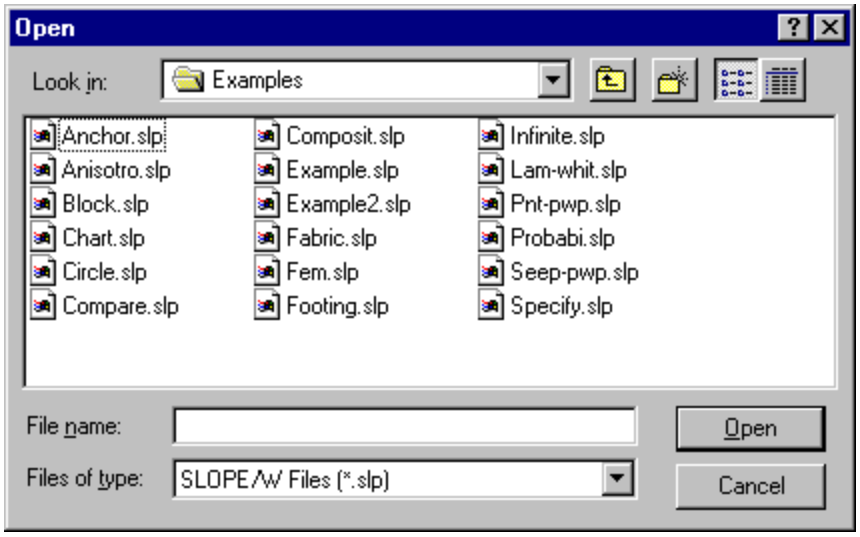
The File menu commands are:

- ***New*** Initializes SOLVE for starting a new analysis. The File New command clears all file names and settings in SOLVE. New has the same action as quitting SOLVE and then restarting SOLVE.
- ***Open Data File*** Selects the DEFINE data file to solve and, if necessary, the data file containing the finite element computed pore-water conditions. For more information about this command, see File Open Data File in this chapter.
- ***Exit*** Quits SLOPE/W SOLVE but does not quit Windows.

File Open Data File

☐ Selects the DEFINE data file to solve.

When you choose File Open Data File, the following dialog box appears:



➢ **To open a file:**

- Type a name in the File Name edit box and then press Open. The file name may include a directory and a path. The file name extension must be omitted or entered as **SLP**.

  -- or --

- Click on a file name in the list box and then press Open.

  -- or --

- Double-click on a file name in the list box.

➢ **To change the current directory or drive:**

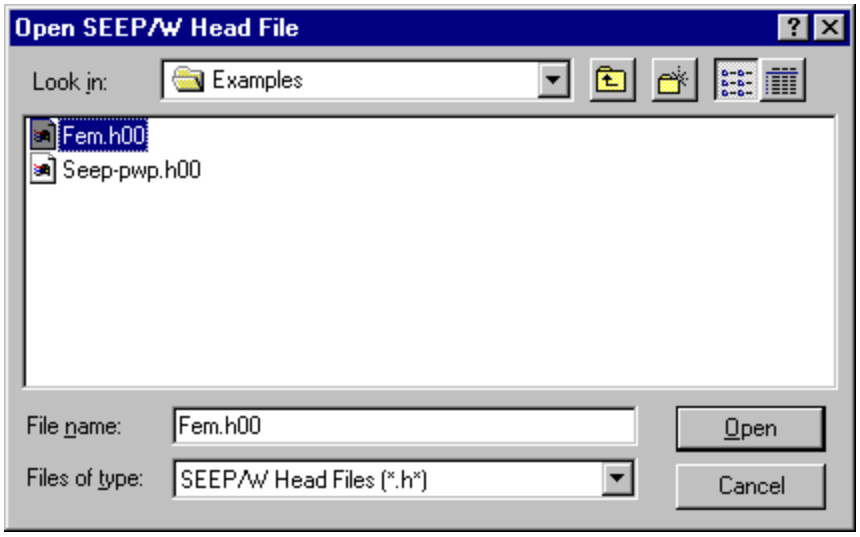
- Use the Look In box to select the drive and directory.

Files Read by SOLVE

Only the specified **SLP** data file is read, and not the secondary **SL2** file. The **SLP** file contains the data required for the factor of safety calculations, while the **SL2** file contains graphical data not required by SOLVE.

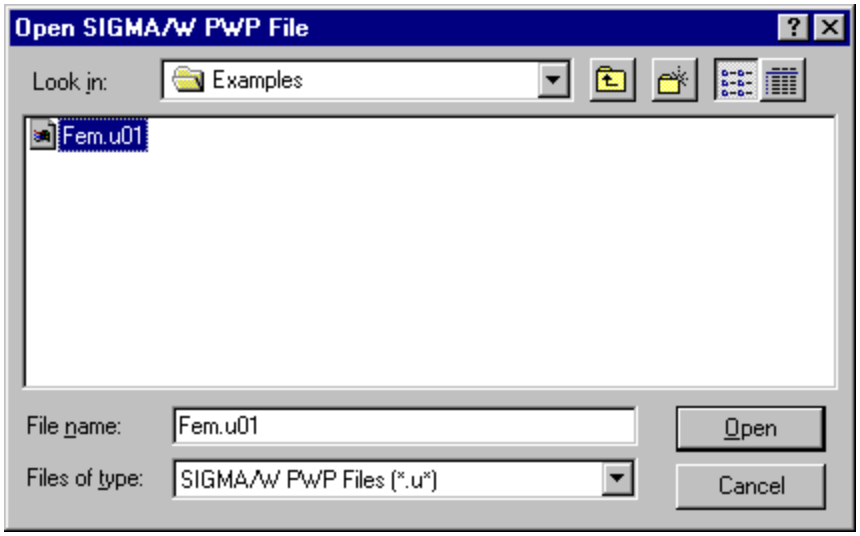
If the data file uses the Finite Element method or if it has the pore-water conditions specified as SEEP/W Heads or SIGMA/W Pressures, SOLVE will prompt you for the additional SEEP/W or SIGMA/W output file names.

If the pore-water conditions are specified as SEEP/W Heads, the following dialog box appears:



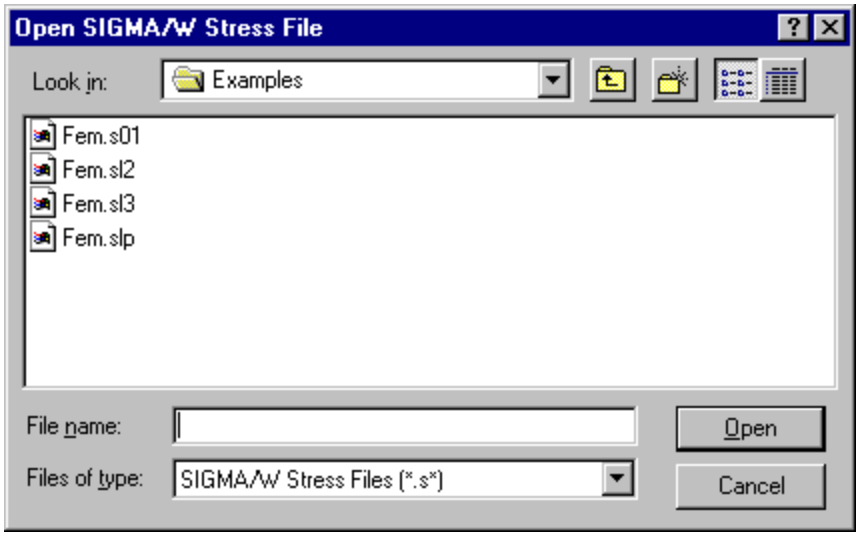
Select and open a SEEP/W computed head file name that has an extension beginning with **H**. The file that you select should correspond to the finite element mesh that was imported into the problem using SLOPE/W DEFINE.

If the pore-water conditions are specified as SIGMA/W Pressures, the following dialog box appears:



Select and open a SIGMA/W computed pore-water pressure file name that has an extension beginning with **U**. The file that you select should correspond to the finite element mesh that was imported into the problem using SLOPE/W DEFINE.

If the problem was analyzed using the Finite Element method, the following dialog box appears:



Select and open a SIGMA/W computed stress file name that has an extension beginning with **S**. The file that you select should correspond to the finite element mesh that was imported into the problem using SLOPE/W DEFINE.

The selected file names are displayed on the SOLVE window.

The Help Menu

The commands available in the SOLVE Help menu operate identically to those available in the DEFINE Help menu. For more information about this menu and its commands, see The Help Menu in Chapter 4.

Running SOLVE

➢ **To run SOLVE:**

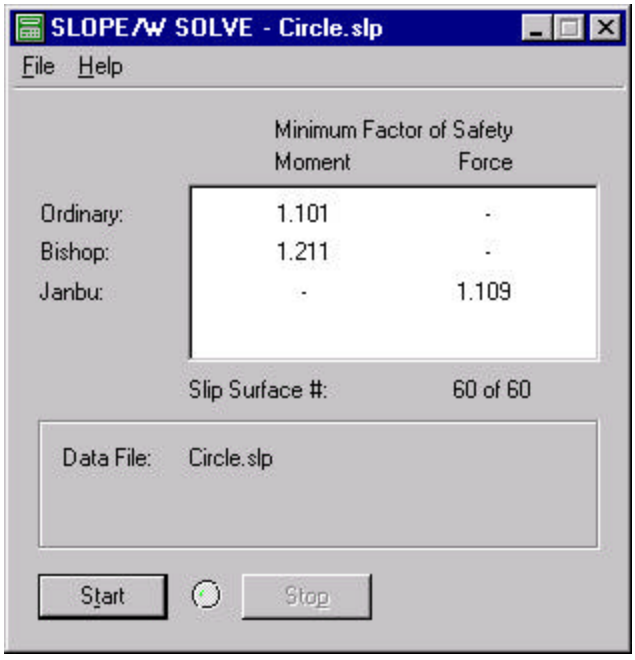
1. Open the DEFINE data file by choosing File Open Data File.

   The data file name appears in the SOLVE window and the Start button becomes active.

2. Click the Start button in the SOLVE window to start processing the solution.

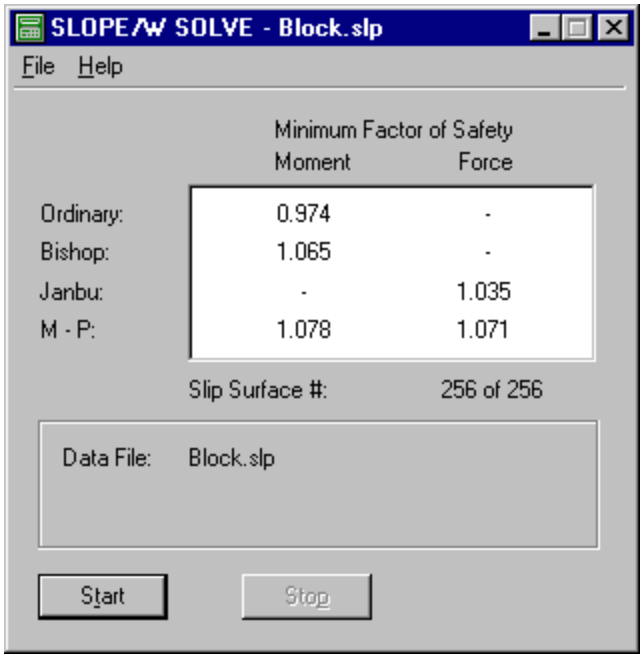
   When the processing starts, the Stop button becomes active and the Start button is grayed. A green dot starts flashing between the Start and Stop buttons. The processing can be halted at any time by clicking the Stop button.

During the processing, the SOLVE window displays the minimum factors of safety and the number of the slip surface being processed. When the processing is finished, the green dot between the Start and Stop buttons stops flashing and a beep is sounded. SLOPE/W finds the factor of safety of all slip surfaces and put the minimum factor of safety of each method on the main window during the computation as illustrated below:



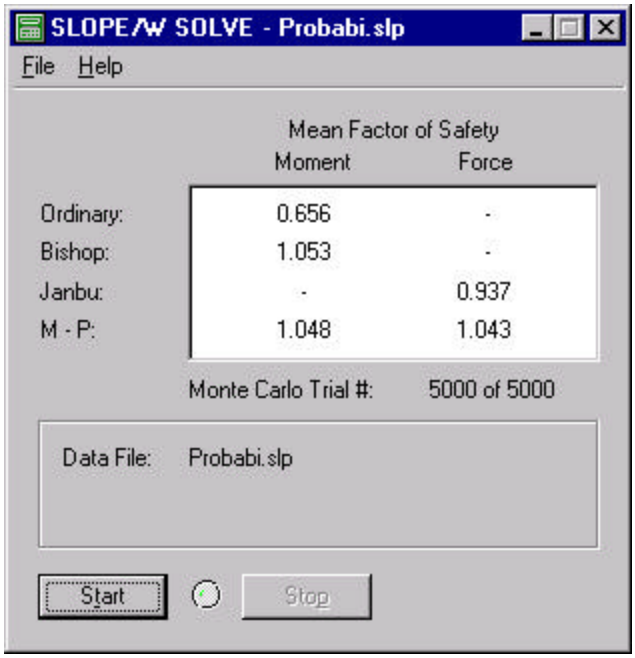
The file name is CIRCLE, and a total of 60 slip surface are solved. The minimum Ordinary, Bishop and Janbu factors of safety are displayed.

A fourth factor of safety is displayed in the SOLVE window if additional analysis method has been selected for the problem in DEFINE. For example, after solving the BLOCK example problem using the Morgenstern-Price method, the SOLVE window appears as follows:



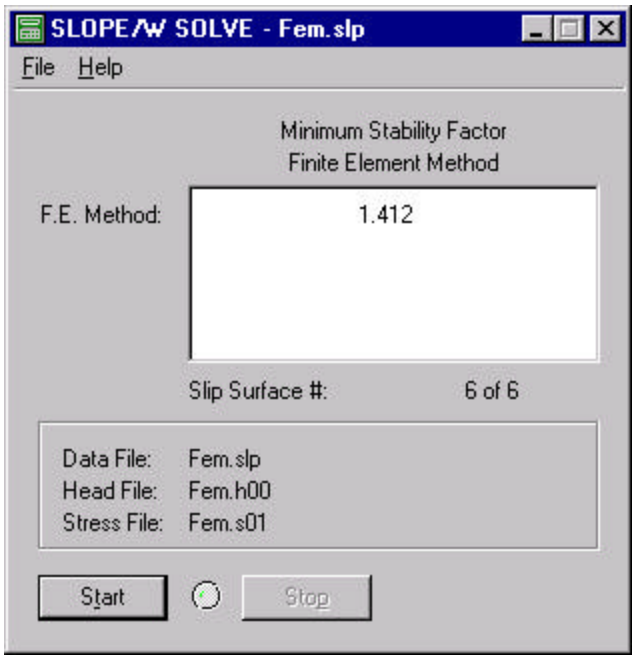
In this BLOCK problem, a total of 256 slip surfaces have been analyzed, and the minimum Ordinary, Bishop, Janbu and Morgenstern-Price factors of safety are displayed.

If probabilistic analysis is required, SLOPE/W continues to find the factor of safety of all Monte Carlo trails on the critical slip surface and put the mean factor of safety of each method on the main window at the end of the probabilistic analysis. For example, after solving the PROBABI probabilistic problem, the SOLVE window appears as follows:



A total of 5000 Monte Carlo trials have been analyzed on the critical slip surface, and the mean Ordinary, Bishop, Janbu and Morgenstern-Price factors of safety of all trials are displayed.

When the Finite Element analysis method is selected in DEFINE, only one minimum factor of safety is displayed in the SOLVE window.



When a solution cannot be found due to convergence problems or invalid slip surfaces, "**No Solution**" is displayed in the SOLVE window instead of the minimum factor of safety. You can still use CONTOUR to view the slip surfaces and determine the reason that no solution was found.

During the computation process, the Stop button is enabled, and you may stop the computation process at any time by pressing the Stop button.

TIP: Once SOLVE starts processing, you can minimize the SOLVE window as an icon, and the processing will take place in the background. This allows you to work with any other Windows application while SOLVE is processing. For example, you can solve several SLOPE/W problems at once, or you can define a new problem while another one is being solved.

Files Created for Limit Equilibrium Methods

For limit equilibrium methods of analysis, the results from the SOLVE analysis are stored in the factor of safety file and the slice forces file. For probabilistic analyses, SOLVE also creates a probability file to store the probabilistic results. All three file names have the same prefix as the problem definition file created by DEFINE. The factor of safety file name has an extension of **FAC**, the slice forces file has an extension of **FRC**, and the probability file name has an extension of **PRO**. The contents of these files are documented in the following sections.

Factor of Safety File - Limit Equilibrium Method

The factor of safety (**FAC**) file contains the computed factors of safety for each slip surface. The following presents a typical factor of safety file:

```
SLOPE/W Example Problem
Slope with a weak clay layer
24/09/97
11:02:54 AM
 3=METHOD   16=NO. OF SLIP SURFACES    1=NO. OF RADII    1=SIDE FUNCTION TYPE
 SLIP     X-         Y-               ITERATION           FACTOR OF SAFETY
  NO.    COORD.     COORD.     RADIUS     NO.    LAMBDA    (MOMENT)   (FORCE)
=============================================================================
   1     30.000     21.000     17.000     1     0.0000      0.928      1.037
   1     30.000     21.000     17.000     4     0.0000      1.305      1.202
   1     30.000     21.000     17.000    29     0.1045      1.328      1.326
   2     32.667     21.000     17.000     1     0.0000      0.855      0.937
   2     32.667     21.000     17.000     4     0.0000      1.153      1.072
   2     32.667     21.000     17.000    27     0.1185      1.178      1.180
   3     35.333     21.000     17.000     1     0.0000      0.863      0.933
   3     35.333     21.000     17.000     4     0.0000      1.136      1.069
   3     35.333     21.000     17.000    28     0.1089      1.157      1.159
   4     38.000     21.000     17.000     1     0.0000      0.946      1.008
   4     38.000     21.000     17.000     5     0.0000      1.222      1.167
   4     38.000     21.000     17.000    34     0.0833      1.237      1.239
   5     30.000     22.333     18.333     1     0.0000      0.939      1.040
   5     30.000     22.333     18.333     5     0.0000      1.274      1.178
   5     30.000     22.333     18.333    20     0.1109      1.294      1.291
   6     32.667     22.333     18.333     1     0.0000      0.870      0.948
   6     32.667     22.333     18.333     5     0.0000      1.136      1.062
   6     32.667     22.333     18.333    18     0.1150      1.155      1.151
   7     35.333     22.333     18.333     1     0.0000      0.877      0.945
   7     35.333     22.333     18.333     5     0.0000      1.127      1.063
   7     35.333     22.333     18.333    18     0.1057      1.144      1.140
   8     38.000     22.333     18.333     1     0.0000      0.941      1.008
   8     38.000     22.333     18.333     5     0.0000      1.199      1.139
   8     38.000     22.333     18.333    21     0.0945      1.213      1.210
   9     30.000     23.667     19.667     1     0.0000      0.951      1.046
   9     30.000     23.667     19.667     5     0.0000      1.253      1.164
   9     30.000     23.667     19.667    16     0.1159      1.270      1.269
  10     32.667     23.667     19.667     1     0.0000      0.891      0.965
  10     32.667     23.667     19.667     5     0.0000      1.135      1.066
  10     32.667     23.667     19.667    15     0.1186      1.150      1.150
  11     35.333     23.667     19.667     1     0.0000      0.893      0.960
  11     35.333     23.667     19.667     5     0.0000      1.125      1.062
  11     35.333     23.667     19.667    16     0.1145      1.140      1.139
  12     38.000     23.667     19.667     1     0.0000      0.941      1.009
  12     38.000     23.667     19.667     6     0.0000      1.182      1.117
  12     38.000     23.667     19.667    17     0.1020      1.195      1.192
  13     30.000     25.000     21.000     1     0.0000      0.966      1.055
  13     30.000     25.000     21.000     5     0.0000      1.239      1.157
```

```
 13     30.000     25.000     21.000    15     0.1177      1.253      1.253
 14     32.667     25.000     21.000     1     0.0000      0.914      0.984
 14     32.667     25.000     21.000     5     0.0000      1.138      1.074
 14     32.667     25.000     21.000    13     0.1161      1.151      1.149
 15     35.333     25.000     21.000     1     0.0000      0.915      0.978
 15     35.333     25.000     21.000     5     0.0000      1.129      1.070
 15     35.333     25.000     21.000    13     0.1106      1.141      1.139
 16     38.000     25.000     21.000     1     0.0000      0.951      1.017
 16     38.000     25.000     21.000     6     0.0000      1.174      1.112
 16     38.000     25.000     21.000    15     0.1095      1.187      1.186

 ---------------------------------------
 | SUMMARY OF MINIMUM FACTORS OF SAFETY |
 ---------------------------------------
 MOMENT EQUILIBRIUM: FELLENIUS OR ORDINARY METHOD
      32.6667=X-COOR.     21.0000=Y-COOR.     17.0000=RADIUS      0.855=F.S.     2=SLIP#
 MOMENT EQUILIBRIUM: BISHOP SIMPLIFIED METHOD
      35.3333=X-COOR.     23.6667=Y-COOR.     19.6667=RADIUS      1.125=F.S.    11=SLIP#
 FORCE EQUILIBRIUM: JANBU SIMPLIFIED METHOD (NO fo FACTOR)
      35.3333=X-COOR.     23.6667=Y-COOR.     19.6667=RADIUS      1.062=F.S.    11=SLIP#
 MOMENT AND FORCE EQUILIBRIUM: MORGENSTERN-PRICE METHOD
      35.3333=X-COOR.     23.6667=Y-COOR.     19.6667=RADIUS      1.140=F.S.    11=SLIP#

 NORMAL TERMINATION OF SLOPE
```

For each slip surface, the **FAC** file lists the number of iterations, the lambda value, and both the moment and force factors of safety.

SOLVE processes each slip surface in three steps:

1. During the first step, no forces are considered between the slices. The resulting moment factor of safety is the Ordinary factor of safety, and the force factor of safety is written in the file only for completeness.

2. For the second step, SOLVE considers normal forces between the slices but with no shear (that is, $l = 0$). SOLVE iterates until the computed factor of safety converge to a specified tolerance.

3. SOLVE proceeds onto the third step if one of the additional methods has been selected that considers a relationship between normal and shear forces between the slices. For example, for the Morgenstern-Price method, SOLVE iterates until the moment and force factors of safety are within the specified convergence tolerance.

Consider the results for Slip Surface 1:

```
SLIP      X-          Y-               ITERATION            FACTOR OF SAFETY
  NO.    COORD.      COORD.      RADIUS      NO.     LAMBDA    (MOMENT)   (FORCE)
================================================================================
   1     30.000      21.000      17.000      1      0.0000      0.928      1.037
   1     30.000      21.000      17.000      4      0.0000      1.305      1.202
   1     30.000      21.000      17.000     29      0.1045      1.328      1.326
```

Ordinary Factor of Safety	= Moment F of S	= 0.928
Bishop Factor of Safety	= Moment F of S	= 1.305
Janbu Factor of Safety	= Force F of S	= 1.202
Morgenstern-Price Factor of Safety	= Moment and Force F of S	= 1.3286

Invalid Factors of Safety

A typical analysis may involve many trial slip surfaces; however, some of the slip surfaces may not have a valid solution. In such cases, a factor of safety larger than 990 is stored in the factor of safety (**FAC**) file. These factors of safety, ranging from 994 to 999, are not really factors of safety; they actually represent different error conditions. CONTOUR interprets the error conditions and displays a message concerning the error when you draw the invalid slip surface using the Draw Slip Surfaces command.

The following summarizes the various error conditions:

- **994** no intersecting point is obtained in the factor of safety versus lambda plot for the GLE method. This happens when an insufficient range of lambda values are specified.
- **995** slip surface could not be analyzed. This happens when the slip surface does not intersect the ground surface or when it only intersects the ground surface in one place.
- **996** inconsistent direction of movement. This happens when the sliding mass described by the slip surface is not in the same direction of movement as the specified direction.
- **997** invalid slip surface grid center. This happens when the slip surface grid center is lower than the entrance or the exit points of the slip surface.
- **998** slip surface could not be analyzed. This happens when the slip surface extends beyond the specified slip surface limits.
- **999** solution cannot converge. This happens when the solution for the slip surface does not converge. Possible factors that contribute to convergence problems are discussed further in Chapter 7.

Minimum Factors of Safety

- SOLVE lists the minimum factors of safety at the end of the **FAC** file, as shown in the above example file listing.
- CONTOUR uses the minimum factors of safety listed at the end of the **FAC** file as the minimum values for each analysis method.

Slice Forces File - Limit Equilibrium Method

The slice forces (**FRC**) file stores the slice forces for the critical slip surface. CONTOUR reads these forces in order to display the free body diagram and force polygon for any slice and to graph various conditions along the critical slip surface.

Quite often, the minimum factor of safety of different methods may not be resulted from the same slip surface. SLOPE/W considers the slip surface that give the minimum factor of safety in the Bishop Simplified method to be the critical slip surface. In the case when an additional method is selected (e.g., the Morgenstern - Price method), the slip surface that give the minimum factor of safety in the additional method is taken to be the critical slip surface.

If the minimum factor of safety is larger than 990 (i.e., no minimum factor of safety could be computed), the critical slip surface is considered suspect and no slice forces file is created.

The following presents a typical slice forces (**FRC**) file:

```
SLOPE/W Verification Problem
Comparison with Lambe and Whitman (1969)
24/09/97
11:19:25 AM

     Center_X       Center_Y       Radius         Slip_Surface   Method
=============================================================================
     1.000000e+001  3.500000e+001  3.000000e+001       1         1

SL#  X_Left         Y_L_Top        Y_L_Bottom     X_Right        Y_R_Top        Y_R_Bottom     Mid_Height
=============================================================================================================
  1  5.507439e+000  5.338293e+000  5.338293e+000  1.000000e+001  8.333333e+000  5.000000e+000  1.751598e+000
```

```
  2  1.000000e+001  8.333333e+000  5.000000e+000  1.270000e+001  1.013333e+001  5.121747e+000  4.202943e+000
  3  1.270000e+001  1.013333e+001  5.121747e+000  1.450000e+001  1.133333e+001  5.339420e+000  5.516550e+000
  4  1.450000e+001  1.133333e+001  5.339420e+000  1.950000e+001  1.466667e+001  6.543893e+000  7.171904e+000
  5  1.950000e+001  1.466667e+001  6.543893e+000  2.450000e+001  1.800000e+001  8.736908e+000  8.828789e+000
  6  2.450000e+001  1.800000e+001  8.736908e+000  2.950000e+001  2.133333e+001  1.220197e+001  9.385080e+000
  7  2.950000e+001  2.133333e+001  1.220197e+001  3.390000e+001  2.426667e+001  1.686744e+001  8.514971e+000
  8  3.390000e+001  2.426667e+001  1.686744e+001  3.500000e+001  2.500000e+001  1.841688e+001  7.017160e+000
  9  3.500000e+001  2.500000e+001  1.841688e+001  3.828427e+001  2.500000e+001  2.500000e+001  3.791178e+000

SL#  L_Load_X     L_Load_Y     A_Load_X     A_Load_Y     P_Load_X     P_Load_Y     A_Modifier
================================================================================================
  1  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  1.0000e+000
  2  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  1.0000e+000
  3  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  1.0000e+000
  4  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  1.0000e+000
  5  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  1.0000e+000
  6  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  1.0000e+000
  7  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  1.0000e+000
  8  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  1.0000e+000
  9  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  1.0000e+000

SL#  Weight       Pore_Water   Alpha        Force Fn.    Seismic_F    Seismic_Y    Pore_Air     Phi_B
=============================================================================================================
  1  9.8365e+002  0.0000e+000 -4.3063e+000  1.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000
  2  1.4185e+003  0.0000e+000  2.5818e+000  1.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000
  3  1.2412e+003  8.1616e+001  6.8953e+000  1.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000
  4  4.4824e+003  1.1105e+003  1.3544e+001  1.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000
  5  5.5180e+003  1.6079e+003  2.3682e+001  1.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000
  6  5.8657e+003  1.5290e+003  3.4722e+001  1.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000
  7  4.6832e+003  6.9961e+002  4.6677e+001  1.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000
  8  9.6486e+002  0.0000e+000  5.4628e+001  1.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000
  9  1.5564e+003  0.0000e+000  6.3486e+001  1.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000

Ordinary_Method_Fm=  1.185     Applied_Lambda=  0.0000
SL#  Normal_M     ShearMob     Phi_Angle    Cohesion
============================================================================
  1  9.8087e+002  8.5988e+002  3.2000e+001  9.0000e+001
  2  1.4171e+003  9.5275e+002  3.2000e+001  9.0000e+001
  3  1.2322e+003  7.4461e+002  3.2000e+001  9.0000e+001
  4  4.3578e+003  2.1038e+003  3.2000e+001  9.0000e+001
  5  5.0533e+003  2.2325e+003  3.2000e+001  9.0000e+001
  6  4.8211e+003  2.1992e+003  3.2000e+001  9.0000e+001
  7  3.2132e+003  1.8138e+003  3.2000e+001  9.0000e+001
  8  5.5854e+002  4.3895e+002  3.2000e+001  9.0000e+001
  9  6.9481e+002  9.2672e+002  3.2000e+001  9.0000e+001

Bishop_Method_Fm=  1.326      Applied_Lambda=  0.0000
SL#  Normal_M     ShearMob     Phi_Angle    Cohesion     SideLeft     ShearLeft    SideRight    ShearRight
=============================================================================================================
  1  1.0465e+003  7.9915e+002  3.2000e+001  9.0000e+001  0.0000e+000  0.0000e+000 -9.7051e+002  0.0000e+000
  2  1.3823e+003  8.3481e+002  3.2000e+001  9.0000e+001  9.7051e+002  0.0000e+000 -1.8329e+003  0.0000e+000
  3  1.1733e+003  6.3742e+002  3.2000e+001  9.0000e+001  1.8329e+003  0.0000e+000 -2.3913e+003  0.0000e+000
  4  4.1787e+003  1.7951e+003  3.2000e+001  9.0000e+001  2.3913e+003  0.0000e+000 -3.3371e+003  0.0000e+000
  5  5.1347e+003  2.0328e+003  3.2000e+001  9.0000e+001  3.3371e+003  0.0000e+000 -3.3362e+003  0.0000e+000
  6  5.5413e+003  2.3041e+003  3.2000e+001  9.0000e+001  3.3362e+003  0.0000e+000 -2.3105e+003  0.0000e+000
  7  4.4780e+003  2.2163e+003  3.2000e+001  9.0000e+001  2.3105e+003  0.0000e+000 -8.2829e+002  0.0000e+000
  8  8.9309e+002  5.4978e+002  3.2000e+001  9.0000e+001  8.2829e+002  0.0000e+000 -4.8857e+002  0.0000e+000
  9  1.2781e+003  1.1028e+003  3.2000e+001  9.0000e+001  4.8857e+002  0.0000e+000 -1.2207e-004  0.0000e+000

Janbu_Method_Ff=  1.186       Applied_Lambda=  0.0000
SL#  Normal_F     ShearMob     Phi_Angle    Cohesion     SideLeft     ShearLeft    SideRight    ShearRight
=============================================================================================================
  1  1.0536e+003  8.9702e+002  3.2000e+001  9.0000e+001  0.0000e+000  0.0000e+000 -9.7051e+002  0.0000e+000
  2  1.3782e+003  9.3096e+002  3.2000e+001  9.0000e+001  9.7051e+002  0.0000e+000 -1.8329e+003  0.0000e+000
  3  1.1651e+003  7.0820e+002  3.2000e+001  9.0000e+001  1.8329e+003  0.0000e+000 -2.3913e+003  0.0000e+000
  4  4.1353e+003  1.9837e+003  3.2000e+001  9.0000e+001  2.3913e+003  0.0000e+000 -3.3371e+003  0.0000e+000
  5  5.0528e+003  2.2291e+003  3.2000e+001  9.0000e+001  3.3371e+003  0.0000e+000 -3.3362e+003  0.0000e+000
  6  5.4090e+003  2.5058e+003  3.2000e+001  9.0000e+001  3.3362e+003  0.0000e+000 -2.3105e+003  0.0000e+000
  7  4.3073e+003  2.3875e+003  3.2000e+001  9.0000e+001  2.3105e+003  0.0000e+000 -8.2829e+002  0.0000e+000
  8  8.4274e+002  5.8803e+002  3.2000e+001  9.0000e+001  8.2829e+002  0.0000e+000 -4.8857e+002  0.0000e+000
  9  1.1563e+003  1.1685e+003  3.2000e+001  9.0000e+001  4.8857e+002  0.0000e+000 -1.2207e-004  0.0000e+000
```

The first table in the **FRC** file lists the analysis method and the center and radius for the critical slip surface:

```
Center_X       Center_Y       Radius         Slip_Surface   Method
=================================================================
     1.000000e+001  3.500000e+001  3.000000e+001       1         1
```

The second table in the **FRC** file lists the computed coordinates of each slice:

```
SL#  X_Left         Y_L_Top        Y_L_Bottom     X_Right        Y_R_Top        Y_R_Bottom     Mid_Height
=============================================================================================================
  1  5.507439e+000  5.338293e+000  5.338293e+000  1.000000e+001  8.333333e+000  5.000000e+000  1.751598e+000
```

X_Left	= *x*-coordinate of the left side of the slice
Y_L_Top	= *y*-coordinate of the upper-left corner of the slice
Y_L_Bottom	= *y*-coordinate of the lower-left corner of the slice
X_Right	= *x*-coordinate of the right side of the slice
Y_R_Top	= *y*-coordinate of the upper-right corner of the slice
Y_R_Bottom	= *y*-coordinate of the lower-right corner of the slice
Mid_Height	= height at the middle of a slice

The third table in the **FRC** file lists the various type of external loads applied to each slice:

```
SL#  L_Load_X     L_Load_Y     A_Load_X     A_Load_Y     P_Load_X     P_Load_Y     A_Modifier
=============================================================================================
  1  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  1.0000e+000
```

L_Load_X	= horizontal component of the line load acting on the slice
L_Load_Y	= vertical component of the line load acting on the slice
A_Load_X	= horizontal component of the anchor load acting on the slice
A_Load_Y	= vertical component of the anchor load acting on the slice
P_Load_X	= horizontal component of the pressure boundary load acting on the slice
P_Load_Y	= vertical component of the pressure boundary load acting on the slice
A_Modifier	= anisotropy modifier factor of the slice obtained from the anisotropic function

The fourth table in the **FRC** file lists the weight, seismic force, pore-water and pore-air forces for each slice:

```
SL#  Weight       Pore_Water   Alpha        Force Fn.    Seismic_F    Seismic_Y    Pore_Air     Phi_B
=============================================================================================================
  1  9.8365e+002  0.0000e+000 -4.3063e+000  1.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000  0.0000e+000
```

Weight	= weight of slice
Pore_Water	= pore-water force acting at base of slice
Alpha	= angle between horizontal and the base of slice
Force_Fn.	= force function used to compute interslice shear force of slice
Seismic_F	= magnitude of horizontal seismic force acting on slice
Seismic_Y	= *y*-coordinate of the point where the seismic load is applied
Pore_Air	= pore-air force acting at base of slice
Phi_B	= angle defining the increase in shear strength for an increase in matric suction

The remaining tables in the **FRC** file list the slice forces associated with the various methods. The computed factor of safety for the critical slip surface and the applied lambda values are also recorded. For example, for the Bishop analysis method, the slice forces are:

```
Bishop_Method_Fm=  1.326       Applied_Lambda=  0.0000
```

```
SL#  Normal_M     ShearMob     Phi_Angle    Cohesion     SideLeft     ShearLeft    SideRight    ShearRight
=============================================================================================================
  1  1.0465e+003  7.9915e+002  3.2000e+001  9.0000e+001  0.0000e+000  0.0000e+000 -9.7051e+002  0.0000e+000
```

Normal_M = base normal force with respect to moment equilibrium

ShearMob = mobilized shear resisting force acting at base of slice

Phi_Angle = frictional angle of the material at base of slice

Cohesion = cohesive force of the material at base of slice

SideLeft = interslice normal force acting at left side of slice

ShearLeft = interslice shear force acting at left side of slice

SideRight = interslice normal force acting at right side of slice

ShearRight = interslice shear force acting at right side of slice

If a valid critical slip surface was found but the factor of safety could not be computed for a secondary analysis method, the slice forces data for the secondary method are not computed and are listed as zero. For example, consider that a valid factor of safety has been computed for the Bishop method, but the corresponding solution for the Janbu method could not converge. The slice forces for the Janbu method are then recorded as zero.

Probability File - Limit Equilibrium Method

The probability (**PRO**) file stores the results from a probabilistic analysis. CONTOUR reads the probability file in order to display the results, the probability density function and the probability distribution function. The following presents a typical probability (**PRO**) file:

```
SLOPE/W Example Problem
Learn Example in Chapter 3
24/09/97
10:36:10 AM


Method     Mean_FOS   SD_FOS     R_Index    F_Prob
=================================================
Ordinary   1.325      0.176      1.848      0.03215
Bishop     1.453      0.181      2.496      0.00626
Janbu      1.313      0.160      1.960      0.02491

Trial #    Ordinary   Bishop     Janbu
=======================================
    1      1.741      1.885      1.690
    2      1.335      1.471      1.332
    3      1.005      1.144      1.037
    4      1.003      1.110      1.014
    5      1.686      1.821      1.641
    6      1.287      1.416      1.281
    7      1.204      1.306      1.186
    8      1.553      1.679      1.521
    9      1.206      1.316      1.210
   10      1.178      1.282      1.167
```

The first table in the **PRO** file summarizes the results of the probabilistic analysis:

Method	= slope stability analysis method
Mean_FOS	= mean factor of safety of all Monte Carlo trials
SD_FOS	= standard deviation on factor of safety
R_Index	= reliability index
F_Prob	= probability of failure

The second table in the **PRO** file lists the computed factor of safety for each method of analysis for all Monte Carlo trials. See Probabilistic Slope Stability Analysis in Chapter 8 for a description of slope stability analyses using the Monte Carlo method.

Files Created for the Finite Element Method

For the Finite Element analysis method, SOLVE stores the analysis results in the factor of safety file and the slice forces file. When probabilistic analyses is requested, SOLVE also creates a probability file to store the probabilistic results. All three file names have the same prefix as the problem definition file created by DEFINE. The factor of safety file name has an extension of **FAC**, the slice forces file has an extension of **FRC**, and the probability file name has an extension of **PRO**.

Factor of Safety File - Finite Element Method

The factor of safety (**FAC**) file contains the computed factors of safety for each slip surface. The following presents a typical factor of safety file when finite element method is used:

```
SLOPE/W User's Guide Example Problem
Finite element method with SIGMA/W stress
24/09/97
11:26:50 AM
 8=METHOD    6=NO. OF SLIP SURFACES    6=NO. OF RADII
 SLIP     X-         Y-                    FACTOR OF SAFETY
  NO.    COORD.     COORD.     RADIUS     FINITE ELEMENT METHOD
================================================================================
  1     65.000     65.000     44.721             1.498
  2     65.000     65.000     46.957             1.412
  3     65.000     65.000     49.193             1.413
  4     65.000     65.000     51.430             1.453
  5     65.000     65.000     53.666             1.516
  6     65.000     65.000     55.902             1.580

 ----------------------------------------
 | SUMMARY OF MINIMUM FACTORS OF SAFETY |
 ----------------------------------------
 FINITE ELEMENT METHOD
      65.0000=X-COOR.     65.0000=Y-COOR.     46.9574=RADIUS      1.412=F.S.     2=SLIP#

 NORMAL TERMINATION OF SLOPE
```

Like the limit equilibrium methods, this files may also store factor of safety larger than 990 for slip surfaces that have no valid solution. These factors of safety, ranging from 994 to 999, are not really factors of safety; they actually represent different error conditions. CONTOUR interprets the error conditions and displays a message concerning the error when you draw the invalid slip surface using the Draw Slip Surfaces command. See, Factor of Safety File - Limit Equilibrium Method in this chapter for a description of the various error conditions regarding Invalid Factors of Safety.

The slip surface that produces the minimum factor of safety is shown at the end of the **FAC** file.

Slice Forces File - Finite Element Method

The slice forces (**FRC**) file stores the slice forces for the critical slip surface. CONTOUR reads these forces in order to display the free body diagram and force polygon for any slice and to graph various conditions along the critical slip surface.

If the minimum factor of safety is larger than 990 (i.e., no minimum factor of safety could be computed), the critical slip surface is considered suspect and no slice forces file is created.

The following presents a typical slice forces file when finite element method is used:

```
SLOPE/W User's Guide Example Problem
Finite element method with SIGMA/W stress
24/09/97
```

```
11:26:50 AM

     Center_X       Center_Y       Radius         Slip_Surface   Method
=============================================================================
     6.500000e+001  6.500000e+001  4.695743e+001       2         8

SL#  X_Left         Y_L_Top        Y_L_Bottom     X_Right        Y_R_Top        Y_R_Bottom     Mid_Height
=================================================================================================================
  1  2.525079e+001  4.000000e+001  4.000000e+001  2.683386e+001  4.000000e+001  3.764410e+001  1.216393e+000
  2  2.683386e+001  4.000000e+001  3.764410e+001  2.841693e+001  4.000000e+001  3.556059e+001  3.427784e+000
  3  2.841693e+001  4.000000e+001  3.556059e+001  3.000000e+001  4.000000e+001  3.369505e+001  5.396809e+000
  4  3.000000e+001  4.000000e+001  3.369505e+001  3.173913e+001  3.913044e+001  3.185314e+001  6.816006e+000
  5  3.173913e+001  3.913044e+001  3.185314e+001  3.347826e+001  3.826087e+001  3.019512e+001  7.692762e+000
  6  3.347826e+001  3.826087e+001  3.019512e+001  3.521739e+001  3.739130e+001  2.869578e+001  8.399170e+000
  7  3.521739e+001  3.739130e+001  2.869578e+001  3.695652e+001  3.652174e+001  2.733618e+001  8.957008e+000
  8  3.695652e+001  3.652174e+001  2.733618e+001  3.869566e+001  3.565217e+001  2.610165e+001  9.382889e+000
  9  3.869566e+001  3.565217e+001  2.610165e+001  4.043479e+001  3.478261e+001  2.498063e+001  9.689810e+000
 10  4.043479e+001  3.478261e+001  2.498063e+001  4.217392e+001  3.391304e+001  2.396379e+001  9.888130e+000

SL#  Pore_Water   Alpha        Sigma_X      Sigma_Y      Tau_XY       Pore_Air     Phi_B        A_Modifier
=================================================================================================================
  1  0.0000e+000  5.6101e+001  1.7843e+001  2.6939e+001  4.5048e+000  0.0000e+000  0.0000e+000  1.0000e+000
  2  0.0000e+000  5.2772e+001  1.4401e+001  6.3953e+001  3.4144e+000  0.0000e+000  0.0000e+000  1.0000e+000
  3  0.0000e+000  4.9683e+001  1.4201e+001  7.9053e+001  3.6647e-001  0.0000e+000  0.0000e+000  1.0000e+000
  4  0.0000e+000  4.6644e+001  2.4164e+001  1.2343e+002 -3.0662e+000  0.0000e+000  0.0000e+000  1.0000e+000
  5  0.0000e+000  4.3632e+001  2.5585e+001  1.4826e+002 -4.7754e+000  0.0000e+000  0.0000e+000  1.0000e+000
  6  0.0000e+000  4.0765e+001  2.6669e+001  1.6405e+002 -8.2585e+000  0.0000e+000  0.0000e+000  1.0000e+000
  7  0.0000e+000  3.8017e+001  3.2675e+001  1.7575e+002 -8.8057e+000  0.0000e+000  0.0000e+000  1.0000e+000
  8  0.0000e+000  3.5369e+001  4.1514e+001  1.9173e+002 -1.0912e+001  0.0000e+000  0.0000e+000  1.0000e+000
  9  9.6567e+000  3.2805e+001  4.4711e+001  2.0144e+002 -1.3375e+001  0.0000e+000  0.0000e+000  1.0000e+000
 10  2.6213e+001  3.0314e+001  4.8276e+001  2.0309e+002 -1.5095e+001  0.0000e+000  0.0000e+000  1.0000e+000

Finite_Element_Method=  1.412
SL#  Normal       Phi_Angle    Cohesion     Strength     ShearMob     Local S.F.
=============================================================================
  1  7.0525e+001  3.0000e+001  1.0000e+001  6.9106e+001  1.6786e+001  4.1170e+000
  2  9.3761e+001  3.0000e+001  1.0000e+001  8.0303e+001  6.4862e+001  1.2381e+000
  3  1.0207e+002  3.0000e+001  1.0000e+001  8.3400e+001  7.8434e+001  1.0633e+000
  4  1.7201e+002  3.0000e+001  1.0000e+001  1.2464e+002  1.2510e+002  9.9634e-001
  5  2.0446e+002  3.0000e+001  1.0000e+001  1.4208e+002  1.4778e+002  9.6138e-001
  6  2.2346e+002  3.0000e+001  1.0000e+001  1.5198e+002  1.5881e+002  9.5696e-001
  7  2.4932e+002  3.0000e+001  1.0000e+001  1.6602e+002  1.5796e+002  1.0511e+000
  8  2.7962e+002  3.0000e+001  1.0000e+001  1.8277e+002  1.5891e+002  1.1501e+000
  9  2.9643e+002  3.0000e+001  1.0000e+001  1.8626e+002  1.5911e+002  1.1706e+000
 10  3.0321e+002  3.0000e+001  1.0000e+001  1.8007e+002  1.5082e+002  1.1939e+000

Number_of_Node_with_Pore_Water_Pressure=   453
Node#       PWP
=============================================================================
    1  2.9421e+002
    2  2.6969e+002
    3  2.4518e+002
    4  2.2066e+002
    5  1.9614e+002
    6  1.7162e+002
    7  1.4710e+002
    8  1.2259e+002
```

The first table in the **FRC** file lists the analysis method, together with the center and radius of the critical slip surface:

```
Center_X       Center_Y       Radius         Slip_Surface   Method
=============================================================================
     6.500000e+001  6.500000e+001  4.695743e+001       2         8
```

The second table in the **FRC** file lists the computed coordinates of each slice:

```
SL#  X_Left         Y_L_Top        Y_L_Bottom     X_Right        Y_R_Top        Y_R_Bottom     Mid_Height
=================================================================================================================
  1  2.525079e+001  4.000000e+001  4.000000e+001  2.683386e+001  4.000000e+001  3.764410e+001  1.216393e+000
```

X_Left	= *x*-coordinate of the left side of the slice
Y_L_Top	= *y*-coordinate of the upper-left corner of the slice
Y_L_Bottom	= *y*-coordinate of the lower-left corner of the slice
X_Right	= *x*-coordinate of the right side of the slice
Y_R_Top	= *y*-coordinate of the upper-right corner of the slice
Y_R_Bottom	= *y*-coordinate of the lower-right corner of the slice
Mid_Height	= height at the middle of the slice

The third table in the **FRC** file lists the stress state at each slice base center, as well as the pore-air and pore-water forces applied to each slice:

```
SL#  Pore_Water   Alpha        Sigma_X      Sigma_Y      Tau_XY       Pore_Air     Phi_B        A_Modifier
=============================================================================================================
  1  0.0000e+000  5.6101e+001  1.7843e+001  2.6939e+001  4.5048e+000  0.0000e+000  0.0000e+000  1.0000e+000
```

Pore_Water	= pore-water force acting at base of slice
Alpha	= angle between horizontal and the base of slice
Sigma_X	= horizontal stress acting at base center
Sigma_Y	= vertical stress acting at base center
Tau_XY	= shear stress acting at base center
Pore_Air	= pore-air force acting at base of slice
Phi_B	= angle defining the increase in shear strength for an increase in matric suction
A_Modifier	= anisotropic modifier factor of the slice obtained from the anisotropic function

The fourth table in the **FRC** file lists the base normal, the soil strength and the local stability factor for each slice:

```
Finite_Element_Method=  1.412
SL#  Normal       Phi_Angle    Cohesion     Strength     ShearMob     Local S.F.
=============================================================================
  1  7.0525e+001  3.0000e+001  1.0000e+001  6.9106e+001  1.6786e+001  4.1169e+000
```

Normal	= base normal force
Phi_Angle	= frictional angle of the material at base of slice
Cohesion	= cohesive force of the material at base of slice
Strength	= resisting shear force at base of slice
ShearMob	= activating shear force at base of slice
Local S.F.	= local stability factor

In cases where the pore-water pressure conditions are specified from a finite element analysis (i.e., from SEEP/W or from SIGMA/W), the pore-water pressure at each node is also presented at the end of the **FRC** file.

Probability File - Finite Element Method

The probability (**PRO**) file stores the results from a probabilistic analysis. CONTOUR reads the probability file in order to display the results, the probability density function and the probability distribution function. The following presents a typical probability (**PRO**) file when finite element method is used :

```
SLOPE/W User's Guide Example Problem
Finite element method
24/09/97
2:18:41 PM


Method     Mean_FOS   SD_FOS     R_Index    F_Prob
=================================================
FEM        1.412      0.097      4.241      0.00001
```

Trial #	FEM
1	1.351
2	1.361
3	1.517
4	1.397
5	1.390
6	1.554
7	1.487
8	1.427
9	1.505
10	1.418

The first table in the **PRO** file summarizes the results of the probabilistic analysis:

Method	= slope stability analysis method
Mean_FOS	= mean factor of safety of all Monte Carlo trials
SD_FOS	= standard deviation on factor of safety
R_Index	= reliability index
F_Prob	= probability of failure

The second table in the **PRO** file lists the computed factor of safety for all Monte Carlo trials. See Probabilistic Slope Stability Analysis in Chapter 8 for a description of slope stability analyses using the Monte Carlo method.

Chapter 6 CONTOUR Reference

Introduction 3

Toolbars 4
- Standard Toolbar 4
- Mode Toolbar 5
- View Preferences Toolbar 6
- Method Toolbar 7

The File Menu 9
- File Open 10

The Edit Menu 12

The Set Menu 13

The View Menu 14
- View Method 14
- View Slice Forces 15
- View Preferences 18
- View Toolbars 22

The Draw Menu 24
- Draw Contours 24
- Draw Contour Labels 25
- Draw Slip Surfaces 26
- Draw Graph 29
- Draw Probability 36

The Sketch Menu 41

The Modify Menu 42

The Help Menu 43

Introduction

SLOPE/W CONTOUR graphically displays the trial slip surfaces and the factors of safety computed by SOLVE. It also provides features for contouring the factors of safety, displaying the detailed forces acting on the minimum slip surface as a free body diagram and force polygon, plotting graphs of conditions along the slip surface from crest to toe, and plotting probability density and distribution functions for probabilistic analyses.

This chapter describes the purpose and operation of each SLOPE/W CONTOUR command. CONTOUR has many features for viewing, labeling, and printing the drawing which are similar to those in DEFINE. This chapter refers you to the appropriate section in Chapter 4 for CONTOUR commands that are identical to DEFINE commands.

All of the CONTOUR commands are accessed by selections from the CONTOUR menu bar or toolbars. The toolbars contain icons which provide a quick way to access many commands available in the menus.

The menus available and the function of each are:

- ***File*** Opens and saves files, imports pictures and prints the drawing. For more information about this command, see The File Menu in this chapter.
- ***Edit*** Copies the drawing to the Clipboard. For more information about this command, see The Edit Menu in this chapter.
- ***Set*** Sets grid, zoom and axes settings. For more information about this command, see The Set Menu in this chapter.
- ***View*** Controls viewing options, displays soil and point information, and displays slice forces as a free body diagram and force polygon. For more information about this command, see The View Menu in this chapter.
- ***Draw*** Draws slip surfaces, factor of safety contours, and graphs of slip surface conditions or probability functions. For more information about this command, see The Draw Menu in this chapter.
- ***Sketch*** Defines graphic objects to label, enhance, and clarify the problem results. For more information about this command, see The Sketch Menu in this chapter.
- ***Modify*** Allows graphic and text objects to be moved or deleted and text objects or pictures to be modified. For more information about this command, see The Modify Menu in this chapter.
- ***Help*** Displays the online help system and information about SLOPE/W. For more information about this command, see The Help Menu in this chapter.

In the remainder of this chapter, the commands in the toolbars and in each of these menus are presented and described.

Toolbars

For general information about toolbars, see Toolbars in Chapter 4.

In CONTOUR, six toolbars are available for performing various tasks as follows:

Standard Toolbar Contains buttons for file operations, printing, copying and redrawing the display. For more information about this toolbar, see Standard Toolbar in this chapter.

Mode Toolbar Contains buttons for entering different operating modes which are used to display and edit graphic and text object data. For more information about this toolbar, see Mode Toolbar in this chapter.

View Preferences Toolbar Contains buttons for toggling various display preferences. For more information about this toolbar, see View Preferences Toolbar in this chapter.

Grid Toolbar Contains controls for specifying the display of a drawing grid. The Grid toolbar in CONTOUR operates identically to the Grid toolbar in DEFINE. For more information about this toolbar, see Grid toolbar in Chapter 4.

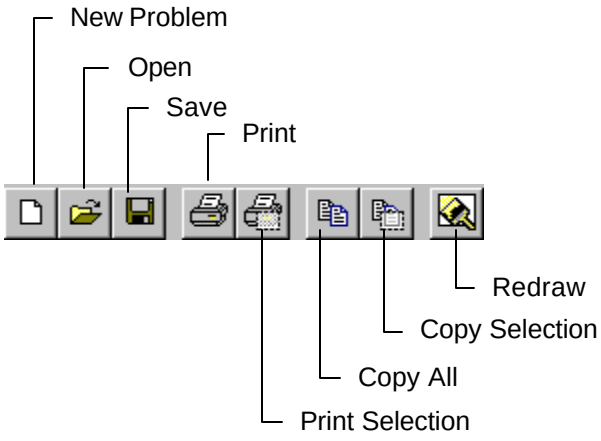
Zoom Toolbar Contains controls for zooming in and out of the drawing. The Zoom toolbar in CONTOUR operates identically to the Zoom toolbar in DEFINE. For more information about this toolbar, see Zoom toolbar in Chapter 4.

Method Toolbar Contains controls used to display the analysis results for a specific slope stability method. For more information about this toolbar, see Method Toolbar in this chapter.

Standard Toolbar

The Standard toolbar, shown in Figure 6.1, contains commands for initializing new problems, opening previously saved problems, saving a current problem's CONTOUR settings, printing the current problem, copying the current problem to the Windows clipboard and redrawing the display.

Figure 6.1 The Standard Toolbar



The toolbar buttons are:

New Problem Use the New Problem button to clear any existing problem definition data and reset the CONTOUR settings back to their defaults. This places the program in the same state as when it was first invoked. This button is a shortcut for the File New command. For more information about is command, see The File Menu in this chapter.

Open Use the Open button as a shortcut for the File Open command. For information about this command, see File Open in this chapter.

Save Use the Save button as a shortcut for the File Save command. For information about this command, see The File Menu in this chapter.

Print Use the Print button as a shortcut for the File Print command. For more information about this command, see The File Menu in this chapter.

Print Selection Use the Print Selection button to print a selected area of the drawing. The Print Selection button operates identically to the Print Selection button on the Standard toolbar in DEFINE. For more information about this button, see Print Selection Button in Chapter 4.

Copy All Use the Copy All button as a shortcut for the Edit Copy All command. For information about this command, see The Edit Menu in this chapter.

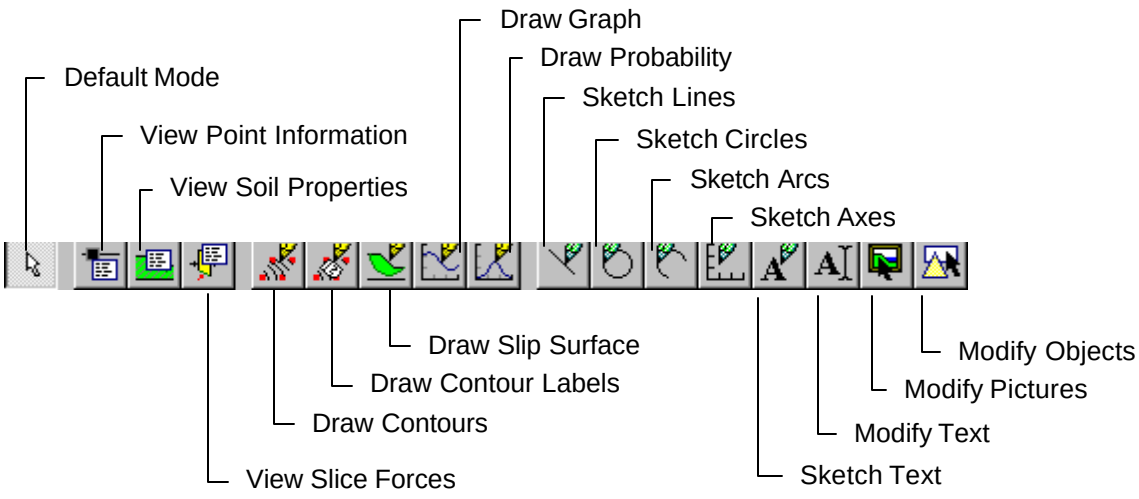
Copy Selection Use the Copy Selection button to copy a selected area of the drawing to the Windows Clipboard. The Copy Selection button operates identically to the Print Selection button on the Standard toolbar in DEFINE. For more information about this button, see Copy Selection Button in Chapter 4.

Redraw Use the Redraw button as shortcut for the View Redraw command. For information about this command, see The View Menu in this chapter.

Mode Toolbar

The Mode toolbar, shown in Figure 6.2, contains buttons that put CONTOUR into "modes" used to accomplish specific tasks such as viewing point and soil information, viewing slice forces, drawing and modifying graphics objects (such as factor of safety contours, contour line labels, slip surfaces, graphs of slip surface conditions, probabilistic graphs and sketch objects), and adding and modifying text and pictures.

Figure 6.2 The Mode Toolbar



The toolbar buttons are:

Default Mode Use the Default Mode button to quit any current mode and return to the default mode.

View Point Information Use the View Point Information button as a shortcut for the View Point Information command. For information about this command, see View Point Information in Chapter 4.

View Soil Properties Use the View Soil Properties button as a shortcut for the View Soil Properties command. For information about this command, see View Soil Properties in Chapter 4.

View Slice Forces Use the View Slice Forces button as a shortcut for the View Slice Forces command. For information about this command, see View Slice Forces in this chapter.

Draw Contours Use the Draw Contours button as a shortcut for the Draw Contours command. For more information about this command, see Draw Contours in this chapter.

Draw Contour Labels Use the Draw Contour Labels button as a shortcut for the Draw Contour Labels command. For more information about this command, see Draw Contour Labels in this chapter.

Draw Slip Surfaces Use the Draw Slip Surfaces button as a shortcut for the Draw Slip Surfaces command. For more information about this command, see Draw Slip Surfaces in this chapter.

Graph Use the Graph button as a shortcut for the Draw Graph command. For more information about this command, see Draw Graph in this chapter.

Draw Probability Use the Draw Probability button as a shortcut for the Draw Probability command. For more information about this command, see Draw Probability in this chapter.

Sketch Lines Use the Sketch Lines button as a shortcut for the Sketch Lines command. For more information about this command, see The Sketch Menu in this chapter.

Sketch Circles Use the Sketch Circles button as a shortcut for the Sketch Circles command. For more information about this command, see The Sketch Menu in this chapter.

Sketch Arcs Use the Sketch Arcs button as a shortcut for the Sketch Arcs command. For more information about this command, see The Sketch Menu in this chapter.

Sketch Axes Use the Sketch Axes button as a shortcut for the Sketch Axes command. For more information about this command, see The Sketch Menu in this chapter.

Sketch Text Use the Sketch Text button as a shortcut for the Sketch Text command. For more information about this command, see The Sketch Menu in this chapter.

Modify Text Use the Modify Text button as a shortcut for the Modify Text command. For more information about this command, see The Modify Menu in this chapter.

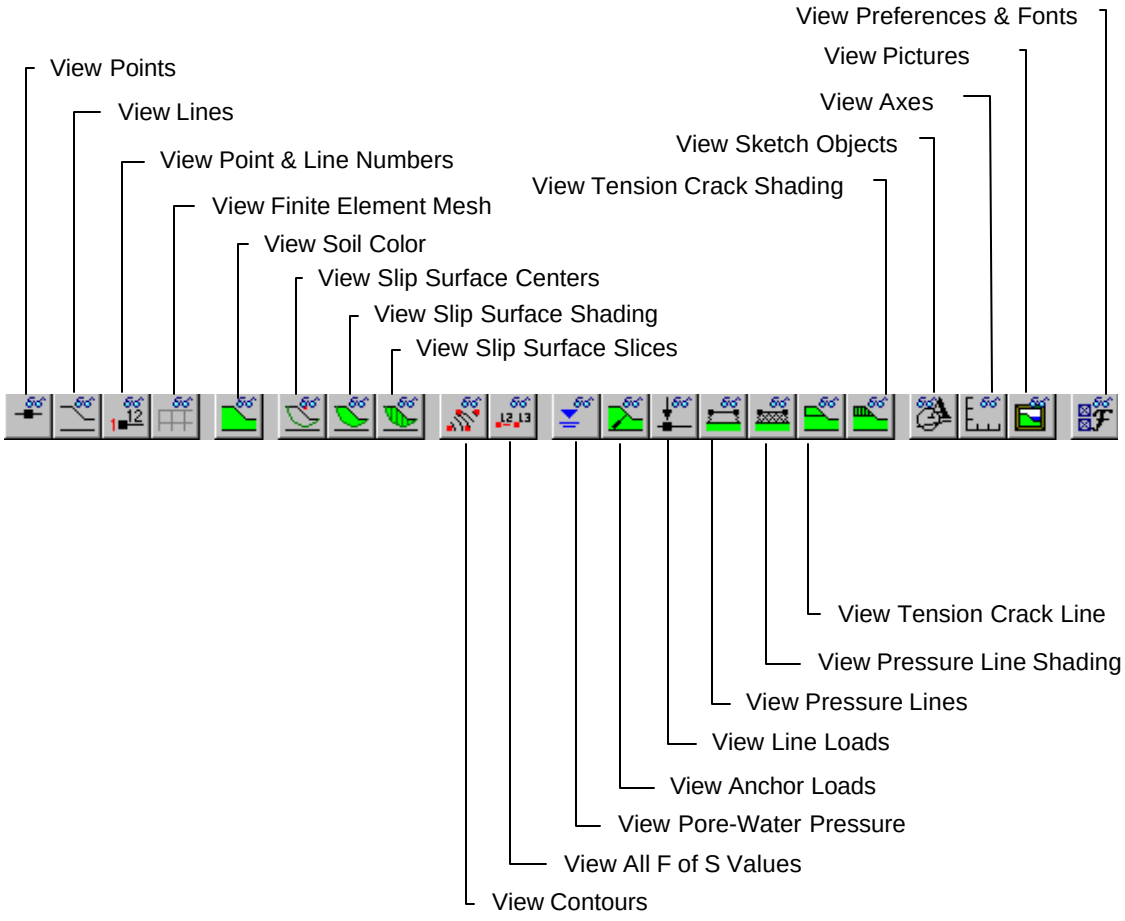
Modify Pictures Use the Modify Pictures button as a shortcut for the Modify Pictures command. For more information about this command, see The Modify Menu in this chapter.

Modify Objects Use the Modify Objects button as a shortcut for the Modify Objects command. For more information about this command, see The Modify Menu in this chapter.

View Preferences Toolbar

The View Preferences toolbar, shown in Figure 6.3, contains buttons for setting viewing preferences such as points and lines and their numbers, soil colors, slip surfaces, pore-water pressure, line and anchor loads, surface pressure lines, tension crack line, sketch objects and text, pictures, and the axes.

Figure 6.3 The View Preferences Toolbar



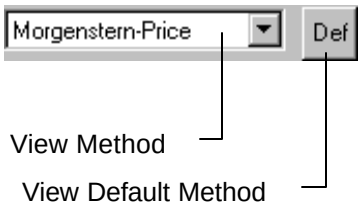
All the buttons on the View Preferences toolbar are shortcuts for the options accessible using the View Preferences command. For more information about this command, see View Preferences in this chapter.

NOTE: Some buttons will only appear on the View Preferences toolbar if the problem requires them. For example, a View Finite Element Mesh button appears only if a finite element mesh was imported into the problem in DEFINE; the View Anchor Loads button appears only if an anchor has been defined.

Method Toolbar

The Method toolbar, shown in Figure 6.4, contains controls used to display the analysis results for a specific slope stability method.

Figure 6.4 The Method Toolbar



The toolbar buttons are:

View Method Use the View Method button to display the slope stability results for a different analysis method. This button is a shortcut for the View Method command.

View Default Method Use the View Default Method button to display the slope stability results for the default analysis method.

The File Menu

The File menu commands are:

- ***New*** Initializes CONTOUR for a new problem. File New clears any existing problem definition data and resets the CONTOUR settings back to their defaults. This command places CONTOUR in the same state as when it was first started.

- ***Open*** Opens and reads existing data files. For more information about this command, see File Open in this chapter.

- ***Import Picture*** Imports a bitmap or metafile into the current drawing. The File Import Picture command in CONTOUR operates the same as the File Import Picture command in DEFINE. For more information about this command, see File Import Picture in Chapter 4.

- ***Export*** Saves drawing in a format suitable for exporting to other programs. The File Export command in CONTOUR operates the same as the File Export command in DEFINE. For more information about this command, see File Export in Chapter 4.

- ***Save*** Saves the current contour drawing information. File Save writes the graphical layout information of the data file name displayed in the CONTOUR window title bar to the **SL3** file. If no problem definition has been opened, this command is disabled.

- ***Save Default Settings*** Saves current settings as default settings. The settings saved include the contour parameters, default font, graph parameters, and view preferences. These settings are used when you open a problem in CONTOUR and choose to not read the **SL3** file.

- ***Print*** Prints the drawing. The File Print command in CONTOUR operates the same as the File Print command in DEFINE. For more information about this command, see File Print in Chapter 4.

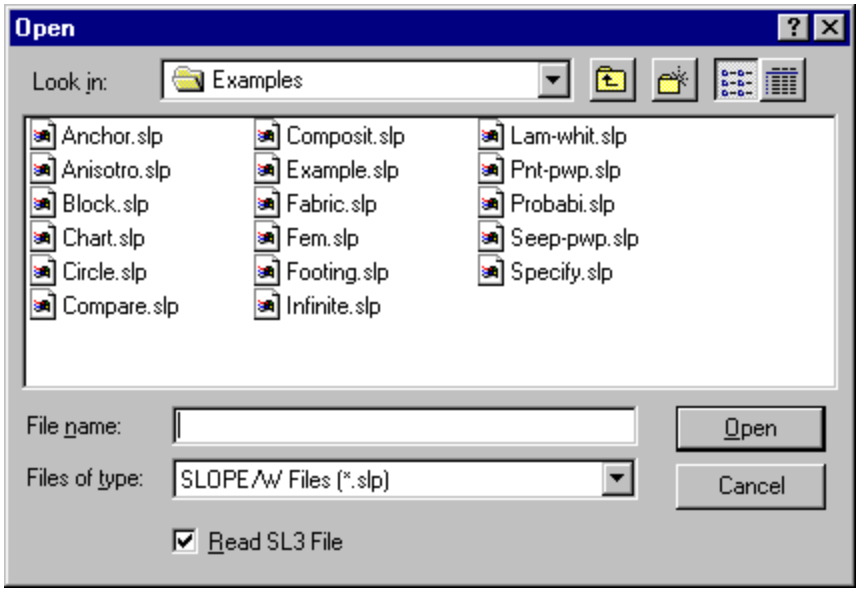
- ***Most Recently Used File*** Allows quick opening of one of the last six files opened. This area of the File menu lists the last six files opened. Selecting a file from the list is a convenient method for opening the file.

- ***Exit*** Quits CONTOUR but does not quit Windows. You are prompted to save the current problem data if any changes have been made.

File Open

☐ Opens and reads existing data files.

When you choose File Open, the following dialog box appears:



➢ **To open a file:**

- Type a name in the File Name edit box and then press Open. The file name may include a directory and a path. The file name extension must be omitted or entered as **SLP**.

  -- or --

- Click on a file name in the list box and then press Open.

  -- or --

- Double-click on a file name in the list box.

➢ **To change the current directory or drive:**

- Use the Look In box to select the drive and directory.

Use the other controls in the dialog box to navigate to the drive and directory containing the SLOPE/W file you wish to open.

NOTE: The SLOPE/W File Open dialog box is a common dialog used by many other Windows applications. To get help on using the dialog box, click on the question-mark in the top-right corner; your cursor then becomes a question mark. Then, click on the dialog control that you need explained; a pop-up window appears with a description of the dialog control. Click anywhere else in the dialog box to remove the pop-up window.

If you check the Read SL3 File check box, SLOPE/W will look for an **SL3** file containing previous CONTOUR settings and, if found, will read the settings. This is normally the preferred option. However, if the **SL3** file contains settings for an earlier problem that has the same file name, you may wish to uncheck the Read SL3 File check box, since the CONTOUR settings (contour values, etc.) may no longer be relevant to the current problem. SLOPE/W will then read the **SL2** file to initialize the page size, scale, default font, etc.

Files Read By CONTOUR

The following files, created by DEFINE, are read when a data file is opened:

- The **SLP** file contains the data required for the slope stability calculations. It is also read by DEFINE and SOLVE.
- The **SL2** file contains information relating to the graphical layout of the problem (e.g., page size and units, engineering units and scale, sketch lines and text, and references to any imported picture files). It is also read by DEFINE, but it is not required by SOLVE.
- The **SL3** file contains the information in the **SL2** file as well as information unique to CONTOUR. It is created by choosing File Save. The **SL3** file is read if it exists and if the Read SL3 File check box is checked; otherwise, the **SL2** file is read.

NOTE: When you open a problem containing imported picture files, SLOPE/W checks to see that the picture file names still exist. If a picture file has been moved or renamed, SLOPE/W displays the Import Picture dialog box, allowing you to specify a different picture file name in its place. See File Import: Picture or Modify Pictures for more information on importing pictures.

The following files, created by SOLVE, are read when a data file is opened:

- The factor of safety file contains the computed factors of safety for each slip surface. The extension of the factor of safety file is **FAC**.
- The slice forces file contains the slice forces for the critical slip surface. The extension of the factor of safety file is **FRC**.
- The probability file contains the results from a probabilistic slope stability analysis. The extension of the probability file is **PRO**.

NOTE: CONTOUR can read data files created by earlier versions of SLOPE/W, including most Version 2 problems. However, CONTOUR will not be able to display as much information for Version 2 data files. It is therefore recommended that you reanalyze any older data files by reading the problem into the latest version of DEFINE, saving it, and reanalyzing it with SOLVE before reading the problem into CONTOUR.

The Edit Menu

The Edit menu commands are:

- ***Copy All*** Copies the entire drawing to the Windows Clipboard. The Edit Copy All command in CONTOUR operates the same as the Edit Copy All command in DEFINE. For more information about this command, see The Edit Menu in Chapter 4.

The Set Menu

The Set menu commands are:

- ***Grid*** Creates a grid of points to assist in drawing objects. The Set Grid command in CONTOUR operates the same as the Set Grid command in DEFINE. For more information about this command, see Set Grid in Chapter 4.
- ***Zoom*** Increases or decreases the size at which the drawing is displayed. The Set Zoom command in CONTOUR operates the same as the Set Zoom command in DEFINE. For more information about this command, see Set Zoom in Chapter 4.
- ***Axes*** Defines scaled reference lines. The Set Axes command in CONTOUR operates the same as the Set Axes command in DEFINE. For more information about this command, see Set Axes in Chapter 4.

The View Menu

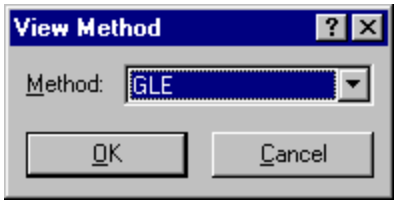
The View menu commands are:

- ***Method*** Identifies the method used for displaying the slope stability results, including factor of safety values and slice forces. For more information about this command, see View Method in this chapter.
- ***Point Information*** Displays information about the selected point. The View Point Information command in CONTOUR operates the same as the View Point Information command in DEFINE. For more information about this command, see View Point Information in Chapter 4.
- ***Soil Properties*** Displays the soil properties for the selected soil or soil line. The View Soil Properties command in CONTOUR operates the same as the View Soil Properties command in DEFINE. For more information about this command, see View Soil Properties in Chapter 4.
- ***Slice Forces*** Displays a free body diagram and force polygon of the forces acting on any slice in the minimum slip surface. For more information about this command, see View Slice Forces in this chapter.
- ***Preferences*** Identifies which items will be displayed on the drawing. For more information about this command, see View Preferences in this chapter.
- ***Toolbars*** Displays or hides the CONTOUR toolbars and the status bar. For more information about this command, see View Toolbars in this chapter.
- ***Redraw*** Redraws the problem. Use the View Redraw command to clear the CONTOUR window and re-display the drawing in the window. This is sometimes needed when drawing objects or when scrolling, since objects may not be completely drawn in the window.

View Method

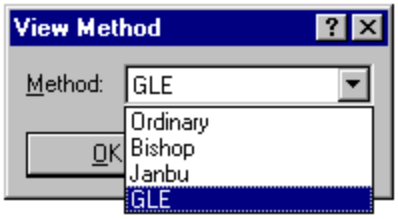
☐ Identifies the method used for displaying the slope stability results, including factor of safety values and slice forces.

When you choose View Method, the following dialog box appears:



➢ **To choose the method for displaying the factor of safety values and the slice forces:**

1. Click on the down-arrow to the right of the Method edit box. A drop-down list box appears, containing a list of the methods for which factors of safety have been calculated.



The Ordinary, Bishop, and Janbu methods always appear in the drop-down list box. If another method has also been used (e.g., GLE), it will be displayed on the last line in the drop-down list box.

2. Select the desired method.

3. Select OK.

   The selected method is displayed in the CONTOUR Method Toolbar. If the minimum slip surface has been displayed with the Draw Slip Surfaces command, CONTOUR displays the minimum slip surface and factor of safety for the selected method. If a selected slip surface has been chosen to view, CONTOUR displays the same slip surface with the factor of safety for the selected method.

Comments

Selecting a method with the View Method command is equivalent to selecting a method in the Method Toolbar.

When a data file is first opened, the method displayed is always the method selected in DEFINE using the KeyIn Analysis Method command.

All CONTOUR commands provide information for the currently-viewed method. For example, the View Slice Forces command displays a free-body diagram and force polygon for the selected method. Select another method in the Method Toolbar and the slice forces will be displayed for the newly-selected method.

View Slice Forces

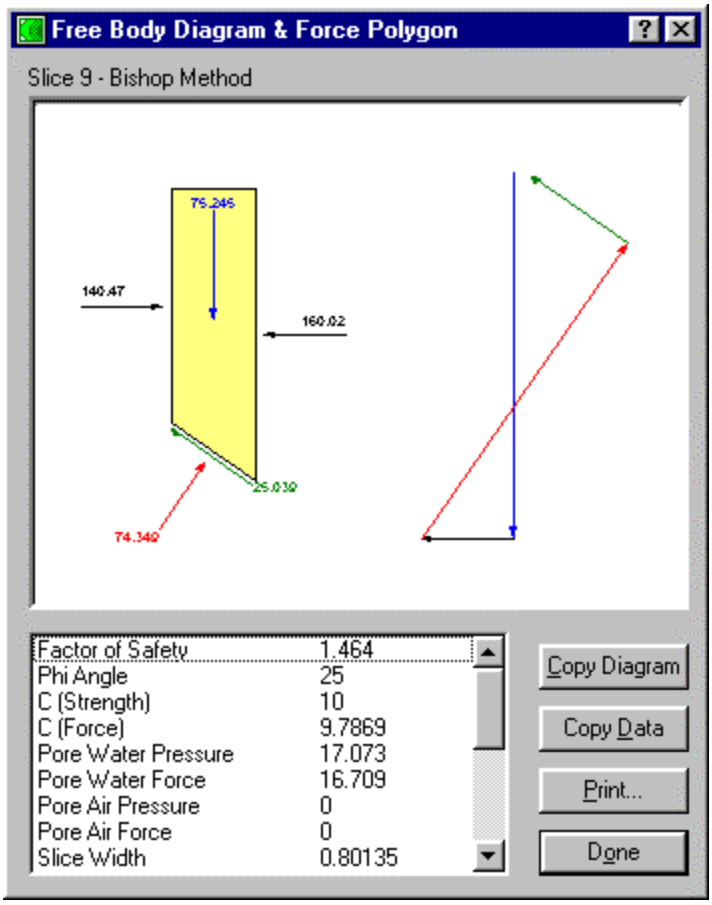
- Displays a free body diagram and force polygon of the forces acting on any slice in the minimum slip surface.

➢ **To view slice forces for the selected method:**

1. Select View Slice Forces from the CONTOUR menu or from the Mode toolbar.

   The cursor changes from an arrow to a cross-hair, the status bar indicates that “View Slice Forces” is the current mode and an empty Force Information dialog box is displayed.

2. Place the cursor at a convenient point within any slice and click the left mouse button. The slice is selected and a free body diagram and force polygon are displayed for the selected slice:



The free body diagram shows the forces on the slice for the selected method. The magnitude of each force vector is displayed beside the arrow (the length of the vectors is not drawn to scale), and the direction of the arrows represents the direction of the vectors.

- The slice weight is displayed as a blue arrow inside the slice, the base normal is displayed as a solid red arrow, and the base shear is displayed as a green arrow parallel to the slice base.
- Interslice normal and shear forces are displayed as black arrows on the sides of the slice.
- The seismic force vector is displayed as a horizontal black arrow at the centroid of the slice; the vector is displayed below its magnitude value if the seismic force does not act on the centroid of the slice.
- The line load force vector is displayed as a black arrow at one of the top corners of the slice.
- The anchor load force vector is displayed as a black arrow above the base of the slice.
- The surface pressure force vector is displayed as a black arrow above the top of the slice and centered in the middle of the slice.
- The force polygon shows the summation of all forces acting on the slice. Closure of the force polygon graphically represents the balance of the slice forces.

The force polygon shows the summation of all forces acting on the slice. Closure of the force polygon graphically represents the balance of the slice forces:

- The weight force is displayed as a blue arrow.
- The side normal forces and seismic force are added together and displayed as a horizontal black arrow.
- The side shear forces are added together and displayed as a vertical black arrow.
- The normal force is displayed as a red arrow.
- The shear force is displayed as a green arrow.
- The line load force is displayed as a black arrow.
- The anchor load force is displayed as a black arrow.
- The surface pressure force is displayed as a black arrow.

The list box displays a list of all the slice forces and other relevant information.

3. To enlarge the free-body diagram and force polygon, drag one of the window corners until the Slice Force Information window is the desired size.

4. Select Copy Diagram to copy the diagram to the Windows Clipboard for use in other Windows applications to create reports, slide presentations, or enhance the diagram.

5. Select Copy Data to copy the listbox slice force information to the Windows clipboard in the following text format:

```
Slice 5
Factor of Safety          1.14
Phi Angle                 30
C (Strength)              20
C (Force)                 28.155
Pore Water Pressure       41.995
Pore Water Force          59.118
Pore Air Pressure         0
Pore Air Force            0
Slice Width               1
Mid-Height                9.5625
Base Length               1.4077
Base Angle                44.736
Polygon Closure Error     2.1516
Anisotropic Strength Modifier    1
Weight                    143.44
Base Shear Force          61.939
Base Normal Force         132.6
Left Side Normal Force    126.02
Left Side Shear Force     14.431
Right Side Normal Force   175.33
Right Side Shear Force    20.079
```

6. Select Print to print the diagram (at the size it is displayed on screen) and/or to print the slice force data.

7. Repeat Steps 2 to 6 until you have finished viewing slice force information. You can move the Slice Force Information window if you need to click on a slice that lies beneath the window.

8. Select Done or click the right mouse button to finish drawing slice forces.

Comments

The free body diagram and force polygon use the minimum slip surface slice forces calculated for the selected method displayed in the Method Toolbar. Select another method in the Method Toolbar or by using the View Method command to display the slice forces for a different method.

SOLVE computes slice forces on the minimum slip surface for the specified method. SOLVE also computes the slice forces for this slip surface using any additional methods (e.g., Bishop, Ordinary, and Janbu). However, this slip surface may not represent the minimum slip surface for the additional method. For example, the Janbu method may compute a different minimum slip surface than the Morgenstern-Price method. If you want to look at the slice forces for the Janbu minimum slip surface, you need to do a separate analysis using only this one particular slip surface.

When you choose View Slice Forces, CONTOUR will display the minimum slip surface if it is not already displayed.

The slice forces along the minimum slip surface can also be plotted using the Draw Graph command.

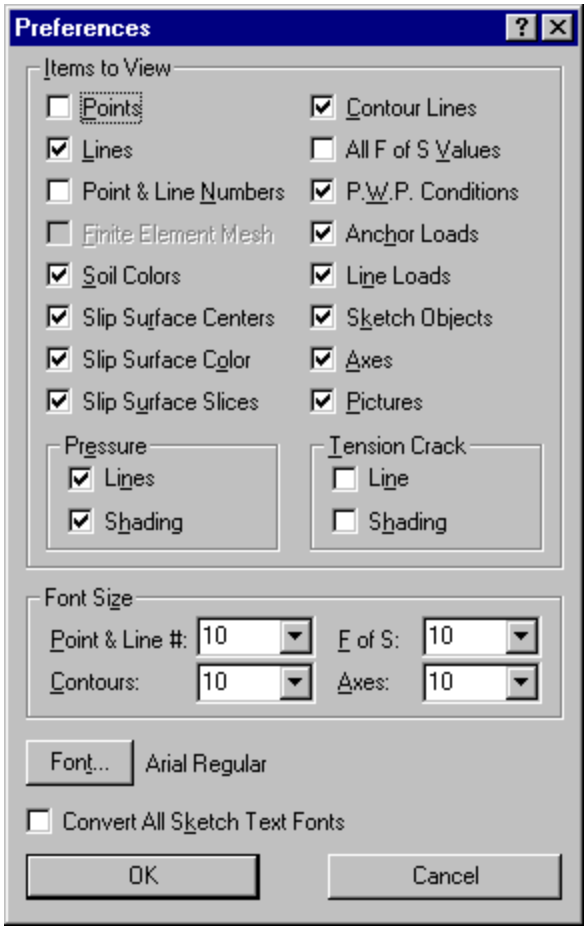
The View Slice Forces command is disabled when the Finite Element Stress analysis method has been used or when you are viewing a problem analyzed with SLOPE/W Version 1.

View Preferences

☐ Identifies which items will be displayed on the drawing.

Use the View Preferences command to select items to view and to change font sizes and the default font.

When you select the Preferences command from the View menu or from the View toolbar, the following dialog box is displayed:



➢ **To select the items to view:**

- In the Items To View group box, check the items that you want displayed on the drawing. Any items that are not checked will not be displayed.

Points Displays points as small squares.

Lines Displays soil geometry lines.

Point & Line Numbers Displays point and line numbers only if points or lines are also displayed.

Finite Element Mesh Displays the finite element mesh imported from a SEEP/W or SIGMA/W data file.

Soil Colors Displays soil layers as different colors, depending on the soil colors defined for each soil type in DEFINE.

Slip Surface Centers Displays the slip surface grid or axis point, an outline of the slip surface, and the factor of safety computed at the currently-viewed slip surface. If this factor of safety represents the minimum for the currently-viewed method, the factor of safety is underlined.

Slip Surface Color Displays the slip surface as a green region.

Slip Surface Slices Displays the slices used in analyzing the slip surface.

Contour Lines Displays factor of safety contour lines on the slip surface grid of centers.

All F of S Values Displays the minimum factor of safety computed at each slip surface center beside the grid center point.

P.W.P. Conditions Displays pore-water pressure conditions. Piezometric lines and contours are displayed as blue dashed lines. Pore-water conditions at points are displayed as triangles. Other conditions, such as r_u values, are not graphically displayed.

Anchor Loads Displays anchor loads as a line segment with an arrow pointing in the direction of the anchor load. The bonded portion of the anchor is shown as a thick line.

Line Loads Displays line loads as small arrows pointing in the direction of the load.

Sketch Objects Displays text, lines, circles, and arcs created by the Sketch command.

Axes Displays the axes.

Pictures Displays imported bitmap or metafile pictures.

Pressure Displays surface pressure lines and/or shading. If Shading is selected, the area between the pressure line and the top soil surface is shaded with a cross-hatch pattern.

Tension Crack Displays the tension crack line and/or shading. If Shading is selected, the area between the tension crack line and the top soil surface is shaded with vertical-line pattern.

Font Sizes

Font sizes for point and line numbers, contour labels, factors of safety, and axes numbers are displayed at the point sizes listed in the Font Size group box.

➢ **To change a font size:**

- Click the down arrow to the right of the Point & Line #, Contours, F of S, or Axes edit boxes and select a point size from the list, or type the desired point size in the edit box.

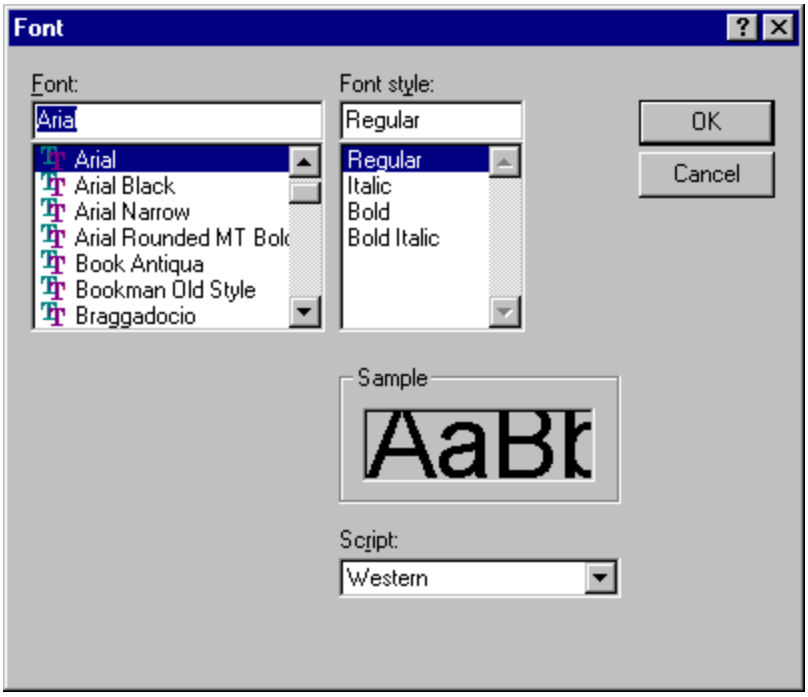
  Points are the units commonly used for font size (72 points is equal to 1 inch). The point size that you enter represents the height of the point, line, contour, or axis numbers at a zoom factor of 1.0.

Default Font

SLOPE/W uses the default font to display point numbers, soil line numbers, axes numbers, axes labels, and graph labels.

➢ **To change the default font:**

1. Click on the Font button. The following dialog box is displayed:



All the fonts that are currently installed in Windows are displayed in the Font list box. To install or delete fonts, you must use the Windows Control Panel. See the Windows documentation for more information on Control Panel.

2. Select the desired font in the Font list box and style in the Font Style list box.

3. Select OK to return to the View Preferences dialog box. The name of the selected font is displayed beside the Font button.

NOTE: SLOPE/W does *not* use the default font to display sketch text on the drawing. Therefore, when you select a new default font, all text defined with the Sketch Text command remains unchanged. This is undesirable if you wish to use one font for all text that appears on the drawing.

➢ **To change the font for *all* sketch text to the default font:**

1. Select the Convert All Sketch Text Fonts check box.

2. When you select the OK button in the View Preferences dialog box, the program asks if you wish to change all sketch text fonts to the default font.

3. Select Yes to change all sketch text fonts to the default font; select No to exit the View Preferences dialog box without changing the sketch text fonts; or select Cancel to return to the View Preferences dialog box.

The Convert All Sketch Text Fonts check box is disabled if there are no sketch text items defined on the drawing.

Comments

Only the items displayed are shown on paper when you print the drawing. This allows you to print any combination of items.

When you define an item, SLOPE/W will check the item in View Preferences if you have not already checked it. For example, if you choose Draw Contours, SLOPE/W will check the Contour Lines option in View Preferences. This enables you to see the contour lines that you define.

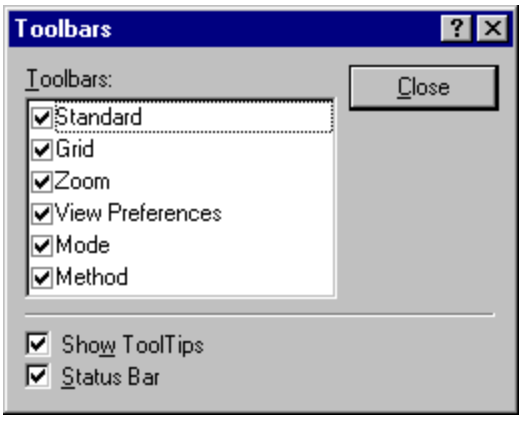
View Toolbars

☐ Displays or hides the CONTOUR toolbars and the status bar.

Use the View Toolbars command to toggle the display of any toolbar, the status bar, or the toolbar tool tips.

➢ **To change the toolbar and status bar display:**

1. Select the Toolbars command from the View menu or right-click on a toolbar and select Toolbars from the pop-up context menu. The following dialog box appears:



2. In the Toolbars list box, check the toolbars you wish to display, or uncheck the toolbars you wish to hide by clicking on the check boxes with the left mouse button.

   Each time you check an item, it appears in the CONTOUR window; each time you uncheck an item, it is removed from the CONTOUR window.

3. To show or remove the tool tips that are displayed when the mouse is over a toolbar button, check or uncheck the Show ToolTips check box.

4. To show or remove the status bar from the bottom of the CONTOUR window, check or uncheck the Status Bar check box. The information displayed in the status bar is described below.

5. When finished, click on the Close button.

NOTE: You can quickly add or remove a toolbar or status bar by clicking the right mouse button on top of any toolbar or status bar. When the pop-up menu appears, select a toolbar or the status bar from the menu to toggle its display.

Status Bar

The status bar contains three panes and is displayed as follows:

Status Information Current status of the program. If the mouse cursor is above a menu item or toolbar button, the purpose of the menu item or toolbar button is displayed. If the program is in a "mode", then the current mode and suggested user action is displayed. The status bar above is shown in the default mode.

Mouse Coordinates Mouse cursor coordinates in engineering units.

The Draw Menu

The function of Draw is to display the slip surfaces and associated factors of safety, to draw and label factor of safety contours, to create free body diagrams and force polygons of slice forces, and to plot graphs of the computed parameter values along the slip surface or graphs of probabilistic results.

The Draw menu commands are:

- ***Contours*** Specifies the factor of safety contours to draw. For more information about this command, see Draw Contours in this chapter.
- ***Contour Labels*** Labels the factor of safety contours. For more information about this command, see Draw Contour Labels in this chapter.
- ***Slip Surface*** Displays the minimum or selected slip surfaces and the associated factors of safety. For more information about this command, see Draw Slip Surfaces in this chapter.
- ***Graph*** Plots graphs of conditions along the minimum slip surface. For more information about this command, see Draw Graph in this chapter.
- ***Probability*** Plots the density and distribution functions computed by a probabilistic slope stability analysis. For more information about this command, see Draw Probability in this chapter.

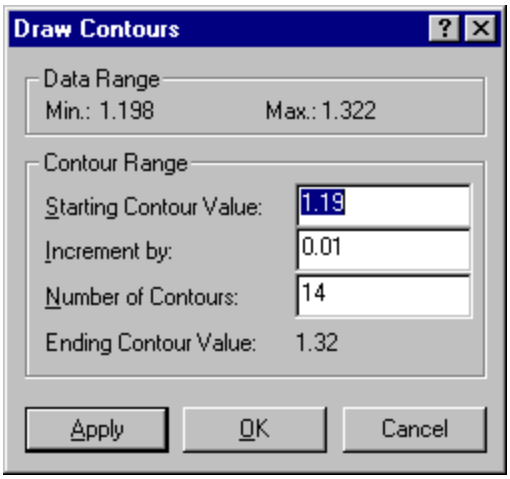
Draw Contours

☐ Specifies the factor of safety contours to draw.

When using the Grid and Radius slip surface option, the minimum factors of safety at each grid center point can be contoured. The Draw Contours command is disabled for problems that use fully-specified or block-specified slip surfaces.

➢ **To draw factor of safety contours:**

1. Choose Draw Contours from the CONTOUR menu or from the Mode toolbar. The following dialog appears:



The Data Range group box displays the minimum and maximum factors of safety for the selected method. Default contour parameters are displayed in the edit boxes and can be used if you want to contour the full range of factors of safety.

2. Edit the settings as necessary in the Contour Range group box.

   Starting Contour Value Specifies the starting, or minimum, contour value (level).

   Increment by Specifies the contour increment.

   Number of Contours Specifies the number of contouring levels. This value must be either a positive number or zero. If it is zero, no contours lines are generated.

   Ending Contour Value Indicates the ending, or maximum, contour value (level). This value depends on the starting contour value, the contour increment and the number of contours.

3. Select the Apply button to see the factor of safety contours generated within the grid of slip surface centers.

4. Repeat Steps 2 to 3 if you wish to change any of the displayed contours.

5. Select OK to accept the contour settings or Cancel to abort.

Comments

To contour the factors of safety for a different analysis method, choose the View Method command or select another method in the Method Toolbar.

To display the minimum factors of safety at each grid center point that are used for contouring, choose the View Preferences command and select the All F of S Values option (you can also adjust the font size of the F of S values as necessary).

Contours are only generated for slip surface grid center points that have a valid computed factor of safety. You can display the error codes for invalid grid points by choosing the Draw Slip Surfaces command and clicking on the invalid grid points.

Draw Contour Labels

☐ Labels the factor of safety contours with contour values.

Use the Draw Contour Labels command to place a label of the contour value at any point on a factor of safety contour line.

➢ **To add contour labels to factor of safety contour lines:**

1. Choose Draw Contour Labels from the CONTOUR menu or from the Mode toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that “Draw Contour Labels” is the current operating mode.

2. Place the cursor at any convenient point on a contour line and click the left mouse button.

   The value of the contour will be displayed on the line.

3. Repeat Step 2 for each contour label you wish to add.

4. Press ESC or click the right mouse button to finish drawing contour labels.

➢ **To delete contour labels from contour lines:**

- Follow the above procedure, except click on an existing contour label, and the label will be removed.

➢ **To change the contour label font:**

- The default font is used to display contour labels. To change the font, choose the View Preferences command and select the Font button or enter a new point size in the F of S Font Size edit box.

Draw Slip Surfaces

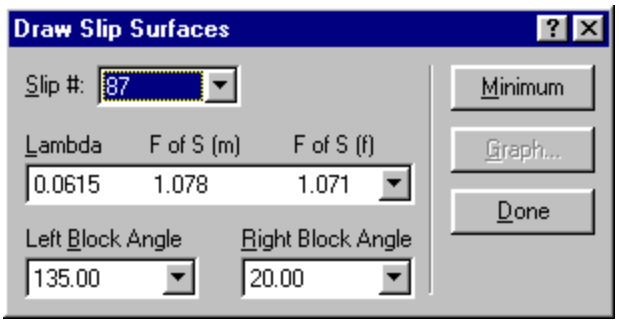
☐ Displays the minimum or selected slip surfaces and the associated factors of safety

When a data file is first opened, CONTOUR displays the minimum slip surface computed for the default analysis method. You can use the Draw Slip Surfaces command to display any of the trial slip surfaces and the associated factors of safety, or you can re-display the minimum slip surface for the currently-selected method.

➢ **To display any of the trial slip surfaces:**

1. Choose Draw Slip Surfaces from the CONTOUR menu or from the Mode toolbar.

   The cursor changes from an arrow to a cross-hair and the status bar indicates that "Draw Slip Surfaces" is the current operating mode. A dialog box similar to the following appears:



   The dialog box displays information about the currently-displayed slip surface: the slip surface number, the Lambda value, and the moment and/or force factors of safety. The Block Angle drop-down list boxes only appear if your problem uses a Block Specified slip surface. The Graph button is disabled unless your problem uses the GLE analysis method.

   NOTE: The lambda value at the intersection of the moment and force Factor of Safety curves tends to be positive when the slope movement is from left to right. When the movement is from right to left, the lambda value at the intersection point tends to be negative.

2. Select the Minimum button to display the minimum slip surface for the currently-viewed analysis method.

   CONTOUR displays the minimum slip surface. If the problem uses the Grid & Radius slip surface option, the factor of safety is displayed beside the slip surface rotation point. For fully specified or block specified slip surfaces, the factor of safety is displayed beside the axis point. The factor of safety is underlined, indicating that this is the minimum factor of safety for the currently-viewed analysis method.

3. To display another Grid & Radius type of slip surface, move the cursor near the rotation center of the desired slip surface and click the left mouse button.

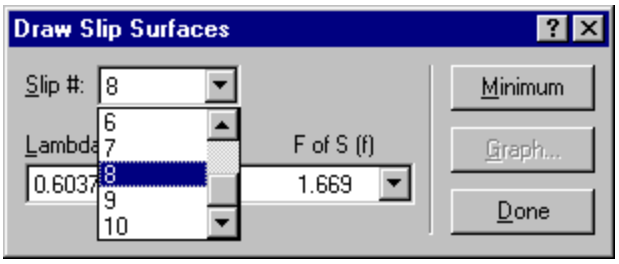
   CONTOUR draws the minimum slip surface for this center and displays the factor of safety beside the grid center point.

4. To display another Block Specified type of slip surface, click on any left or right block intersection point.

   CONTOUR draws the minimum slip surface for the selected left and right block intersection points; the factor of safety is displayed beside the axis point.

To see the other slip surfaces for these block intersection points, select another angle from the Left or Right Block Angle drop-down list boxes.

5. To display another Fully Specified type of slip surface or to display other Grid & Radius slip surfaces with the same grid center, click the down arrow to the right of the Slip # edit box to display a drop-down menu of other slip surfaces. Click on one of the listed slip surface numbers to view the slip surface:



The selected slip surface and its factor of safety are displayed in the CONTOUR window. The dialog box displays the new slip surface number, its Lambda value, and its moment and/or force factors of safety.

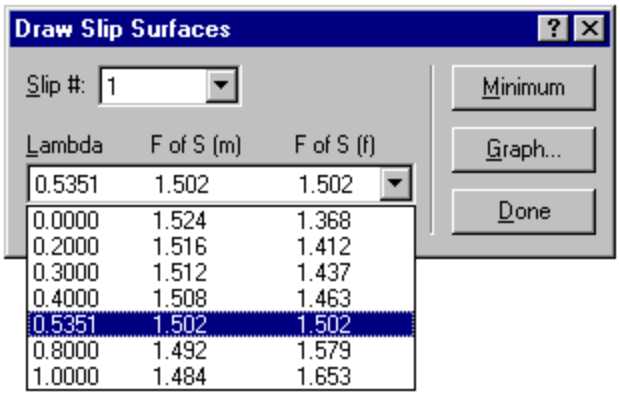
6. Repeat Steps 2 to 5 for all slip surfaces to view.

7. Select Done, press ESC or click the right mouse button to finish viewing slip surfaces.

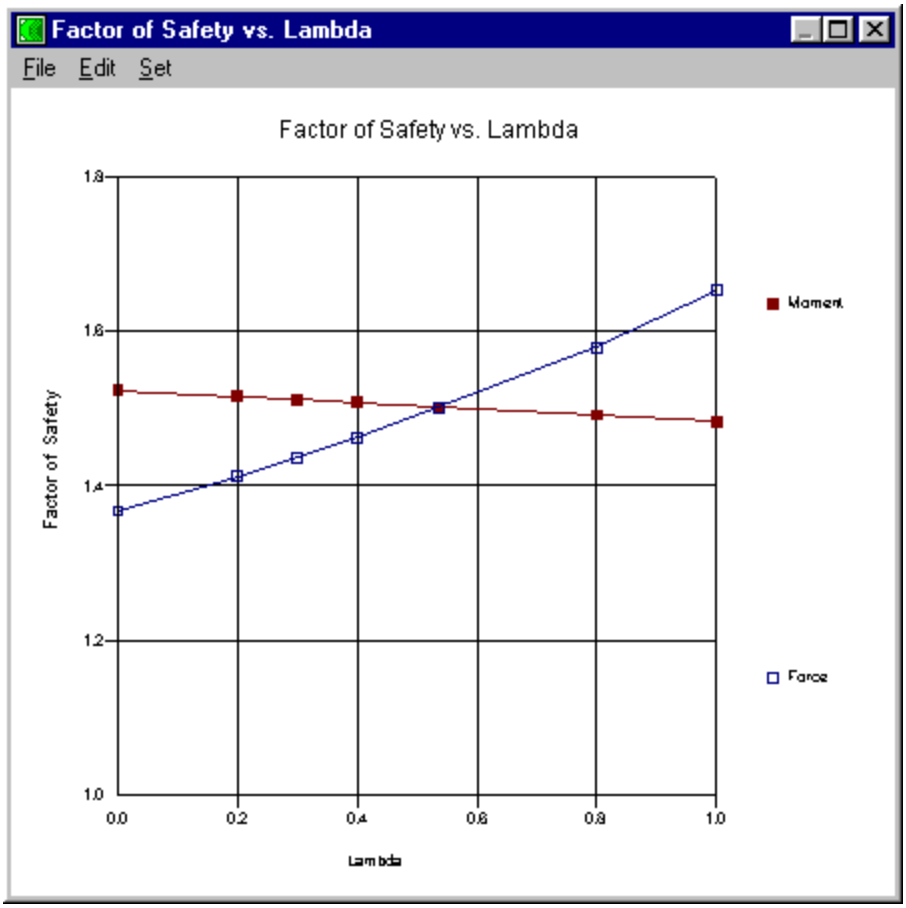
If the GLE analysis method is currently viewed, you can also plot factor of safety versus lambda for both the moment and force factors of safety.

➢ **To create the plot of factor of safety versus lambda for GLE analyses:**

1. In the Draw Slip Surfaces dialog box, click the down arrow to the right of the box containing the lambda values and factors of safety. A drop-down menu of the moment and force factors of safety at each lambda value is displayed.



2. Select the Graph button to plot the listed moment and force factors of safety versus lambda. The following window is displayed:



The window can be moved by dragging the window title bar with the mouse. It can be enlarged by clicking the Maximize button in the upper-right corner of the window.

3. Select any other slip surface to draw. The Graph window will be updated with the moment and force factors of safety for the newly selected slip surface.

The Graph window contains a menu with the following commands:

- ***File Print*** Prints the graph on the selected printer.

NOTE: File Print displays a common Print dialog used by many other Windows applications. To get help on using the dialog box, click on the question-mark in the top-right corner; your cursor then becomes a question mark. Then, click on the dialog control that you need explained; a pop-up window appears with a description of the dialog control. Click anywhere else in the dialog box to remove the pop-up window.

- ***File Close*** Closes the Graph window and returns to the Draw Slip Surfaces dialog box.
- ***Edit Copy*** Copies the graph to the Windows Clipboard for use in other Windows applications. See Edit Copy All in Chapter 4 for further information on copying to the clipboard.
- ***Set Options*** Specifies the options to use when displaying the graph. See the Draw Graph command reference for further information on changing the graph display options.

➢ **To view the minimum slip surface for a different analysis method:**

1. Select the analysis method to view in the Method Toolbar.

   A new factor of safety value, corresponding to the selected method, is displayed for the current slip surface.

2. Select the Minimum button in the Draw Slip Surfaces dialog box to display the minimum slip surface for the newly-selected method.

Invalid Factors of Safety

If SOLVE was unable to compute a factor of safety for the selected slip surface, an error condition, ranging from E994 to E999, is displayed in the Draw Slip Surfaces dialog box. The following summarizes the various error conditions that may be displayed:

- **E994** no intersecting point is obtained in the factor of safety versus lambda plot for the GLE method. This happens when an insufficient range of lambda values are specified.
- **E995** slip surface could not be analyzed. This happens when the slip surface does not intersect the ground surface or when it only intersects the ground surface in one place.
- **E996** inconsistent direction of movement. This happens when the sliding mass described by the slip surface is not in the same direction of movement as the specified direction.
- **E997** invalid slip surface grid center. This happens when the slip surface grid center is lower than the entrance or the exit points of the slip surface.
- **E998** slip surface could not be analyzed. This happens when the slip surface extends beyond the specified slip surface limits.
- **E999** solution cannot converge. This happens when the solution for the slip surface does not converge. Possible factors that contribute to convergence problems are discussed in Chapter 7, Modelling Guidelines.

If SOLVE was unable to compute a factor of safety for a selected slip surface using the GLE method, you may still be able to graph lambda vs. factor of safety. No factor of safety was found because SOLVE was unable to compute the lambda value at which the moment and force factors of safety were equal. To overcome this problem, specify a different range of lambda values using the KeyIn Analysis Control command in DEFINE.

Draw Graph

☐ Plots graphs of conditions along the minimum slip surface.

The Draw Graph command allows you to produce plots of conditions along the slip surface from crest to toe. The following parameters can be plotted for all analysis methods except the Finite Element Stress method:

- Strength (Cohesive, Frictional, and Suction)
- Shear Resistance (Shear Strength and Shear Mobilized)
- Base Cohesion
- Base Friction Angle
- Base Phi B
- Base Normal Stress
- Pore-Water Pressure

- Pore-Air Pressure
- Interslice Force Fn. (Applied Fn. and Specified Fn.)
- Interslice Forces (Normal and Shear Forces)
- Weight per Slice Width
- Seismic Force per Slice Width
- m_a

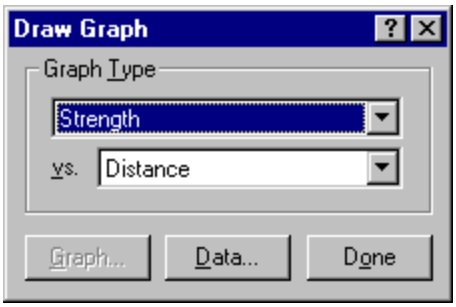
When the Finite Element Stress analysis method has been used, the following parameters can be plotted:

- Strength (Cohesive, Frictional, and Suction)
- Shear Resistance (Shear Strength and Shear Mobilized)
- Base Cohesion
- Base Friction Angle
- Base Phi B
- Base Normal Stress
- Pore-Water Pressure
- Pore-Air Pressure
- Local Stability Factor for each slice
- Sigma X
- Sigma Y
- Tau XY

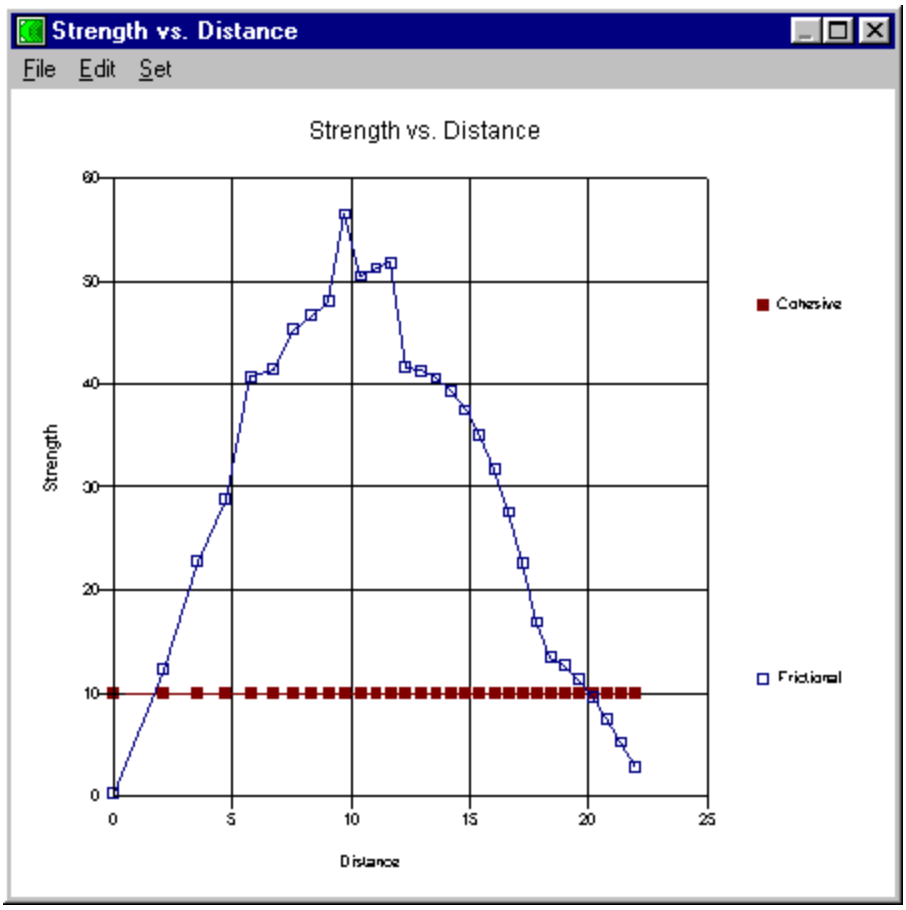
Draw Graph also can be used to extract data values for all slices on the minimum slip surface. These values can be saved as an ASCII text file or copied to the Windows Clipboard and then taken into other Windows graphing applications (e.g., Microsoft Excel).

➢ **To draw a graph of the computed conditions along the slip surface:**

1. Choose Draw Graph from the CONTOUR menu or from the Mode toolbar. The following dialog box appears:



The following Graph window also appears, containing a graph of the selected conditions:



2. In the first drop-down list box in the Graph Type group box, select any of the computed conditions along the slip surface that you wish to plot.

3. In the second drop-down list box in the Graph Type group box, select the way in which to plot along the slip surface. You can plot the selected condition versus the distance along the base of each slice, the *x*-coordinate of the base of each slice, or the slice number.

   When a new parameter is selected in the Graph Type group box, a new graph is plotted in the Graph window.

4. Repeat Steps 2 to 3 for each graph that you wish to display.

The Graph window contains a menu with the following commands:

- ***File Print*** Prints the graph on the selected printer.

  NOTE: File Print displays a common Print dialog used by many other Windows applications. To get help on using the dialog box, click on the question-mark in the top-right corner; your cursor then becomes a question mark. Then, click on the dialog control that you need explained; a pop-up window appears with a description of the dialog control. Click anywhere else in the dialog box to remove the pop-up window.

- ***File Close*** Closes the Graph window and returns to the Draw Graph dialog box.

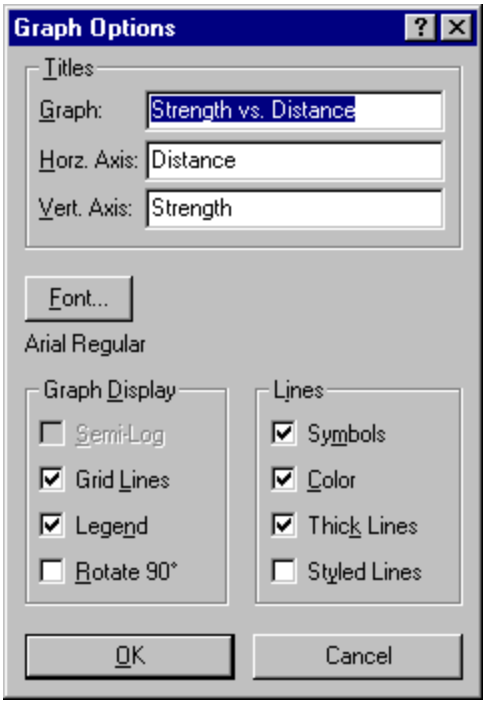
- ***Edit Copy*** Copies the graph to the Windows Clipboard for use in other Windows applications. See Edit Copy All in Chapter 4 for further information on copying to the clipboard.

- ***Set Options*** Specifies the options to use when displaying the graph.

Changing the Graph Display

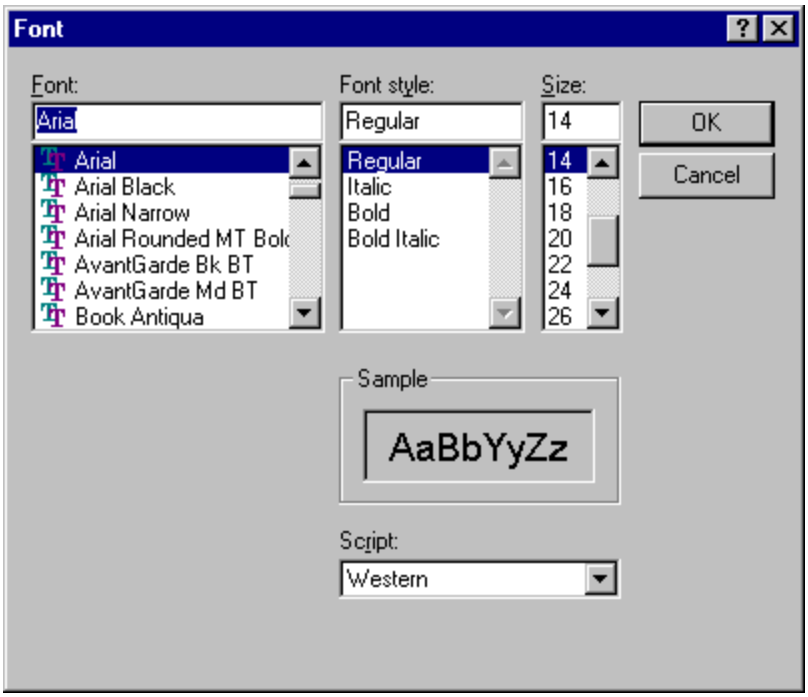
➢ **To specify the graph display options:**

1. Choose Set Options from the Graph window menu. The following dialog box appears:



2. To change the titles, type a new graph title or axis title in the edit boxes.

3. To change the font, select the Font button. The following dialog box appears:



All the fonts that are currently installed in Windows are displayed in the Font list box. To install or delete fonts, you must use the Windows Control Panel. See the Windows documentation for more information on Control Panel.

4. Select the desired font in the Font list box and style in the Font Style list box.

5. Select a font size from the Size list box or type the desired font size in the Size edit box.

   The font size units are relative to the size of the Graph window (i.e., whenever the Graph window is enlarged, the text in the window is also enlarged). Select a font size that results in the graph titles being displayed at a suitable size.

6 Select OK to return to the Set Graph Options dialog box. The name of the selected font is displayed underneath the Font button.

7. To change the graph display options, check any of the following check boxes in the Graph Display group box:

   - ***Semi-Log*** Displays the vertical axis at a log scale. This option is not available if any of the values along the vertical axis are negative or equal to zero.
   - ***Grid Lines*** Displays background grid lines on the graph.
   - ***Legend*** Displays a legend describing each line on the graph
   - ***Rotate 90°*** Plots the independent variable along the vertical axis and the dependent variable along the horizontal axis. This is the default option when the independent variable is the nodal *y*-coordinates.

8. To specify how the lines are plotted on the graph, check any of the following check boxes in the Lines group box:

   - ***Symbols*** Displays symbols at each point on each graph line.

- ***Color*** Displays the lines and symbols on the grid in color.
- ***Thick Lines*** Displays each graph line as a thick line. This option cannot be used in combination with Styled Lines.
- ***Styled Lines*** Displays each graph line as a styled (dashed or dotted) line. This option cannot be used in combination with Thick Lines.

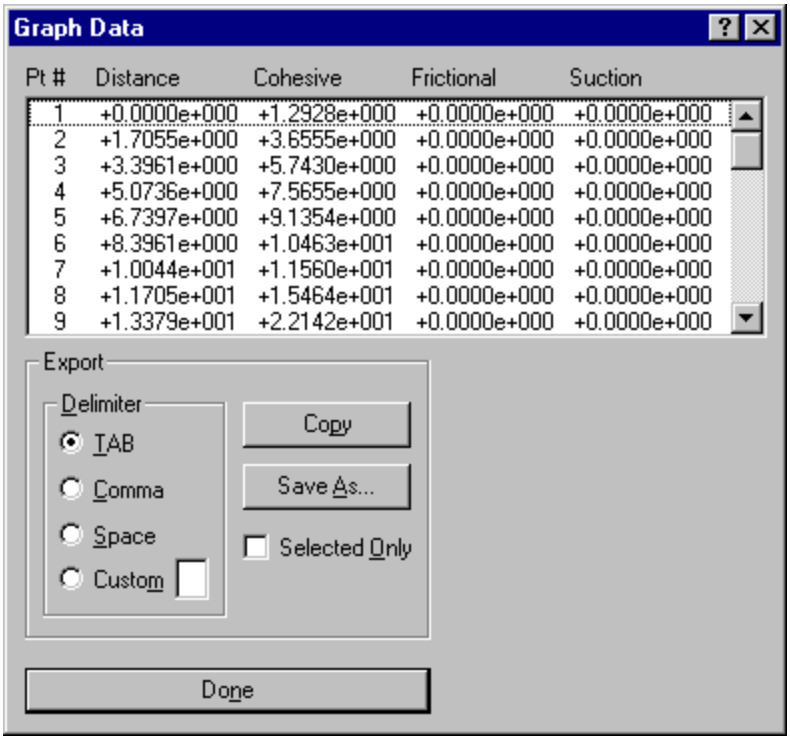
9. Select OK when you have finished selecting the graph display options. The graph is redrawn using the new options.

Extracting the Graph Data

Draw Graph also gives you access to the data used in plotting the graph. This allows you to use the results computed by SOLVE in other applications (e.g., word processors, spreadsheets, or graphing applications) for presentation purposes.

➢ **To access the data used in plotting the graph:**

1. In the Draw Graph dialog box, select the Data button. The following dialog box appears, containing a list of the data used to plot the graph in the Graph window:



The remaining steps describe how to export the data in the list box to the Windows Clipboard or as an ASCII text file.

2. In the Export Delimiter group box, select the character to use as the field delimiter between list box columns.

   Many spreadsheets and databases use a special character to separate data into fields. For example, to import the graph data into Microsoft Excel, select the TAB character. If your application uses a delimiting character that is not listed in the group box, select Custom and type the character in the adjacent edit box.

3. To export a portion of the graph data displayed in the list box, check the Selected Only check box and select the desired lines in the list box.

   A group of lines can be selected either by pressing the CTRL key and clicking on each line in the group or by pressing the SHIFT key and clicking on the first and last line in the group.

   If Selected Only is not checked, the entire list box will be exported.

4. To copy the list box contents to the Windows Clipboard, select the Copy button.

   A beep is sounded when the data points have been copied to the clipboard.

5. To export the list box contents to an ASCII text file, select the Save As button. The following dialog box appears:

NOTE: The File Save As dialog box is a dialog used by many other Windows applications. To get help on using the dialog box, click on the question-mark in the top-right corner; your cursor then becomes a question mark. Then, click on the dialog control that you need explained; a pop-up window appears with a description of the dialog control. Click anywhere else in the dialog box to remove the pop-up window.

6. Type the name you wish to give the file and select the directory in which to save the file.

7. Select OK to export the graph data to the specified ASCII file.

   The Graph Data dialog box is redisplayed when the file has been saved.

8. Select Done in the Graph Data dialog box when you are finished extracting data.

Comments

The graph uses the minimum slip surface conditions calculated for the currently-selected method. To graph slip surface conditions for a different analysis method, choose the View Method command or select another method in the Method Toolbar.

SOLVE computes conditions on the minimum slip surface for the specified method. SOLVE also computes the conditions for this slip surface using any additional methods (e.g., Bishop, Ordinary, and Janbu). However, this slip surface may not represent the minimum slip surface for the additional method. For example, the Janbu method may compute a different minimum slip surface than the Morgenstern-Price method. If you want to graph the conditions for the Janbu minimum slip surface, you need to do a separate analysis using only this one particular slip surface.

When you choose Draw Graph, CONTOUR will display the minimum slip surface if it is not already displayed.

Choose View Slice Forces to display a free body diagram and force polygon of the forces on any slice in the minimum slip surface.

Draw Probability

☐ Plots the density and distribution functions computed by a probabilistic slope stability analysis.

The Draw Probability command allows you to display the results of a probabilistic analysis by plotting either a probability density function or a probability distribution function.

Probability Density Function

The probability density function consists of the following:

- ***# of classes*** The resulting factors of safety from all Monte Carlo trials are grouped into the specified number of classes; each class represents a factor of safety range. The Normal Curve and Histogram functions plot one point for every class. Each class has the same size; the size is calculated by dividing the factor of safety range (e.g., 0.95 to 1.45 gives a range of .5) by the number of classes (e.g., 20). Therefore, increasing the number of classes results in a smaller class size and more data points on the function plots.

- ***Normal Curve*** The results from all Monte Carlo trials are used to generate a normal distribution function over the entire range of computed factors of safety. Each data point on the curve represents the average factor of safety within the corresponding class. For more information on the Normal Curve, see the Normal Distribution Function section in the Theory chapter.

- ***Histogram*** The histogram is a bar graph representing the number of Monte Carlo trials that fall within each class. Each bar represents the number of Monte Carlo trials (as a percentage) that fall within the corresponding class (i.e., factor of safety range).

Probability Distribution Function

The probability distribution function consists of the following:

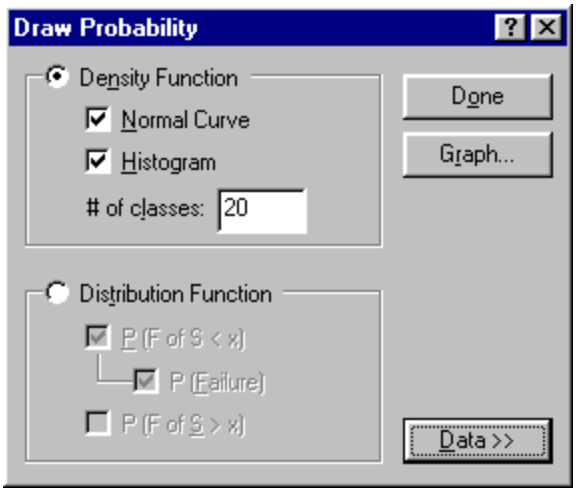
- ***P (F of S < x)*** For a given factor of safety, this function shows the probability that the computed factor of safety is less than the given value.

- ***P (Failure)*** This line on the P(F of S < *x*) function indicates the probability of slope failure. It shows the probability (in percentage) of the factor of safety being less than 1.0.

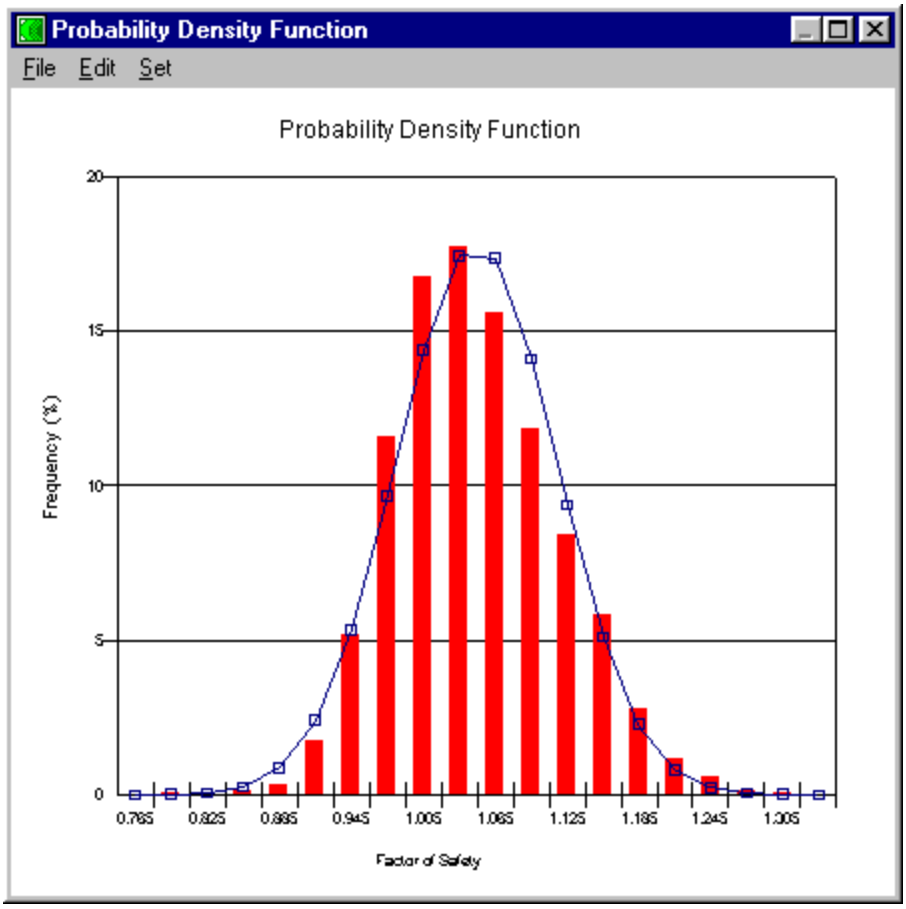
- ***P (F of S > x)*** For a given factor of safety, this function shows the probability that the computed factor of safety is greater than the given value.

➢ **To display probabilistic results:**

1. Choose Draw Probability from the CONTOUR menu or from the Mode toolbar. The following dialog box appears:



The selected probability function also appears (e.g., the Probability Density Function):

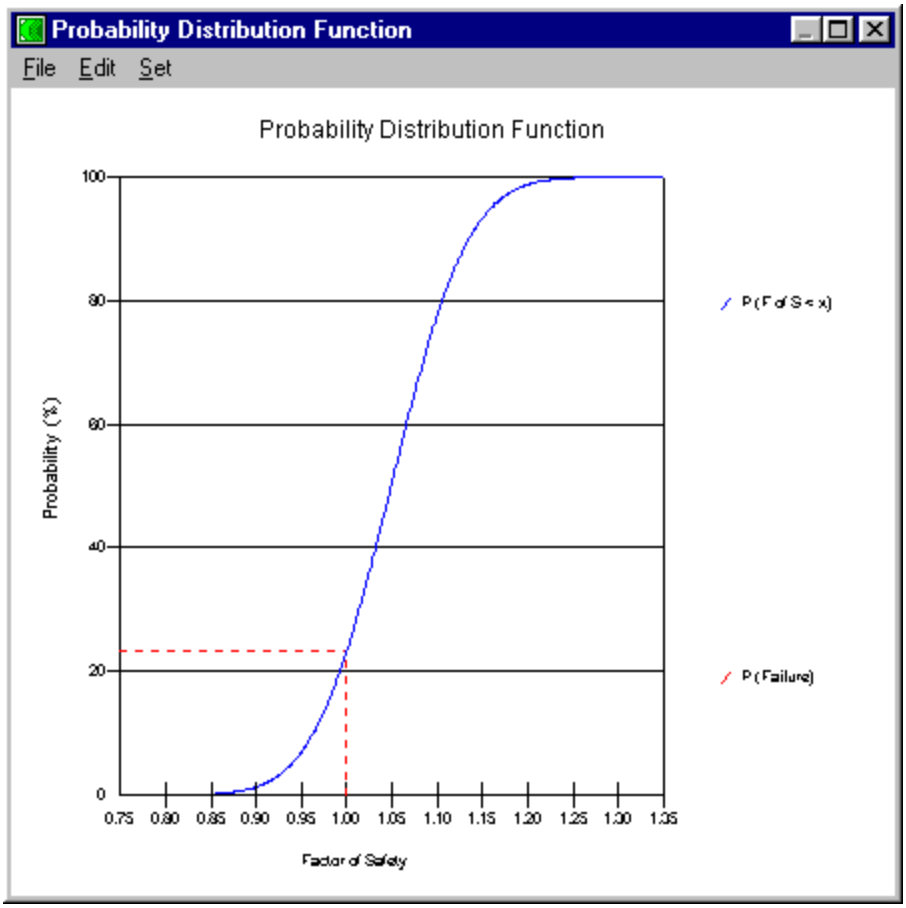


The Normal Curve and/or Histogram are shown on the Probability Function if they are selected in the dialog box.

2. To change the number of points used to compute the density function, type in a new # of classes in the Density Function group box and press the Refresh button.

3. Select the Normal Curve or Histogram options to display or hide these functions on the plot.

4. Select the Distribution Function option to display the Probability Distribution Function as follows:

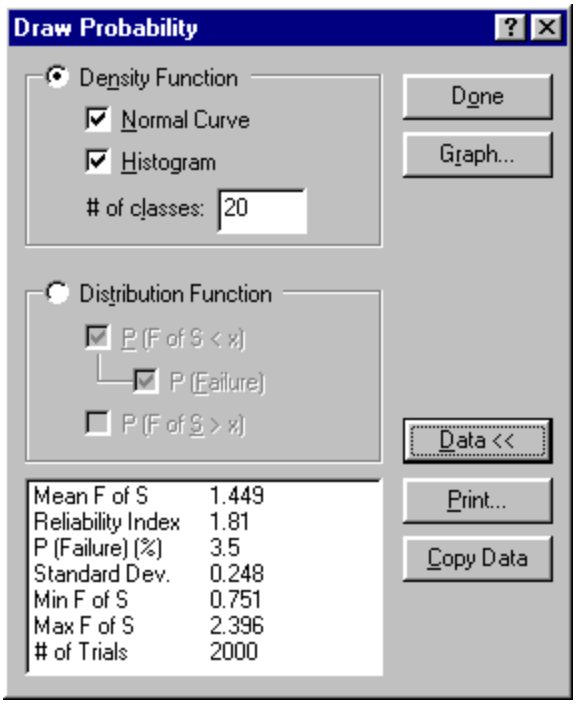


The P(F of S < *x*) function and the P(Failure) line are shown if they are selected in the dialog box.

5. Select the P(F of S > *x*) option to also plot this function in the window.

6. To display a list of the computed probabilistic data, select the Data<< button.

The dialog box enlarges, showing the computed data in a list box as follows:



You can print the data by pressing the Print button, or you can copy the data to the Windows Clipboard by pressing the Copy Data button.

7. To hide the list of computed probabilistic data, select the Data<< button.

8. Select Done when you are finished viewing the probabilistic results.

The Probability Function window contains a menu with the following commands:

- ***File Print*** Prints the function on the selected printer.

NOTE: File Print displays a common Print dialog used by many other Windows applications. To get help on using the dialog box, click on the question-mark in the top-right corner; your cursor then becomes a question mark. Then, click on the dialog control that you need explained; a pop-up window appears with a description of the dialog control. Click anywhere else in the dialog box to remove the pop-up window.

- ***File Close*** Closes the Probability Function window and returns to the Draw Probability dialog box.

- ***Edit Copy*** Copies the Probability Function to the Windows Clipboard for use in other Windows applications. See Edit Copy All in Chapter 4 for further information on copying to the clipboard.

- ***Set Options*** Specifies the options to use when displaying the function. See the Draw Graph command reference for further information on changing the function display options.

Comments

The Probability Function uses the probabilistic results calculated for the currently-selected method. To plot probabilistic conditions for a different analysis method, choose the View Method command or select another method in the Method Toolbar.

When you choose Draw Probability, CONTOUR will display the minimum slip surface if it is not already displayed.

For a more detailed discussion of SLOPE/W probabilistic analyses, see the Probabilistic Slope Stability Analysis section in the Theory chapter.

The Sketch Menu

The commands available in the CONTOUR Sketch menu operate similarly to those available in the DEFINE Sketch menu. For more information about this menu and its commands, see The Sketch Menu in Chapter 4.

The Modify Menu

The commands available in the CONTOUR Modify menu operate similarly to those available in the DEFINE Modify menu. For more information about this menu and its commands, see The Modify Menu in Chapter 4.

The Help Menu

The commands available in the CONTOUR Help menu operate identically to those available in the DEFINE Help menu. For more information about this menu and its commands, see The Help Menu in Chapter 4.

Chapter 7 Modelling Guidelines

Introduction 7-3

Modelling Progression 7-4

Units 7-5

Selecting Appropriate X and Y Coordinates 7-6

Adopting a Method 7-7

Effect of Soil Properties on Critical Slip Surface 7-10

Steep Slip Surfaces 7-11

Weak Subsurface Layer 7-12

Seismic Loads 7-13

Geofabric Reinforcement 7-16

Structural Elements 7-17

Active and Passive Earth Pressures 7-18

Partial Submergence 7-20

Complete Submergence 7-21

Right-To-Left Analysis 7-22

Pore-Water Pressure Contours 7-23

Finite Element Stress Method 7-24

Probabilistic Analysis 7-25

Introduction

SLOPE/W is a powerful tool for modelling the stability of earth structures, and can be used to analyze a wide variety of problems and conditions. However, the problem analyzed must be realistic; that is, the potential mode of failure must be physically admissible. SLOPE/W cannot judge the physical admissibility of a potential failure mode. This is the responsibility of the user. Attempting to analyze physically inadmissible failure modes can cause numerical problems, and SLOPE/W may be unable to compute a factor of safety. Selecting unrealistic soil properties can also create numerical difficulties with computing a factor of safety. Therefore, some planning and judgment is required to use SLOPE/W effectively.

This chapter presents some general modelling guidelines. The information presented is not an exhaustive statement on the "how-to" of modelling. Instead, it is intended to provide suggestions on how you might model various conditions, as well as to outline the implications and consequences of certain modelling specifications.

Modelling Progression

Many modelling difficulties can be overcome by progressing from the simple to the complex. It is good practice to initially define a simplified version of the problem and then add complexity in stages.

Moving from the simple to the complex makes it easier to pinpoint difficulties with the model when the results of the analysis are unrealistic. Determining what causes unrealistic results can be difficult if all of the possible complexities are included at the start of the analysis. Furthermore, it is important that the results obtained are of a form similar to results obtained from simple hand calculations. It is easier to make this judgment if you start with a simplified version of the problem.

The principle of moving from the simple to the complex can be illustrated by considering the effect of anchor loads or seismic loads. Such loads are often applied in a horizontal or near-horizontal direction, which can have dramatic effects on the factor of safety calculations. To be certain of the effect, anchor loads should be applied in increments. The change in factor of safety should be gradual and in a common direction. A gradual, smooth change suggests that the loads are being applied correctly.

Units

Any set of units can be used in a SLOPE/W analysis. However, the units must be consistent throughout the analysis.

Units must be selected for length, force, and unit weight. The unit weight of water is set when the units of length are selected. Table 7.1 shows examples of consistent sets of units.

Table 7.1 Examples of Consistent Units

Property	Units	Example of Metric Units	Example of Imperial Units
Geometry	L	meters	feet
Unit Weight of Water	F/L^3	kN/m^3	p.c.f.
Soil Unit Weight	F/L^3	kN/m^3	p.c.f.
Cohesion	F/L^2	kN/m^2	p.s.f.
Water Pressure	F/L^2	kN/m^2	p.s.f.
Pressure Head	L	meters	feet.
Line Load	F/L	kN/m	lbs/ft.

Selecting Appropriate X and Y Coordinates

Consideration must be given in the selection of the starting *x*- and *y*-coordinates (datum) of a problem. Using a large starting *x*- or *y*-coordinate may affect the precision of the computed results due to round-off error. SOLVE performs most of its computations using single precision floating point values, which carry between 6 and 7 significant digits of precision. Round-off error occurs when a small number, such as 0.123456, is added to a large number, say 100000. If these values are added together, the summed value is 100000.123456. However, a single precision computation in SOLVE stores the summed value as 1.00000E5. As a result, the small number in the summation would be rounded off completely during the addition operation. In many cases, round-off error will also affect the convergence of a problem. Since there are many computations in SOLVE involving the additions and subtractions of *x*- and *y*-coordinates, it is important to minimize unnecessary round-off error due to the poor selection of datum. The easiest way to minimize round-off error is ensure that both the starting *x*- and *y*-coordinates are as close to zero as possible.

During the verification process, DEFINE determines the *x*- and *y*-coordinate ranges of all the points and issues a warning message if more than one significant digit of precision will be lost due to floating point round-off error. For more information about the verification process, see Tools Verify in Chapter 4.

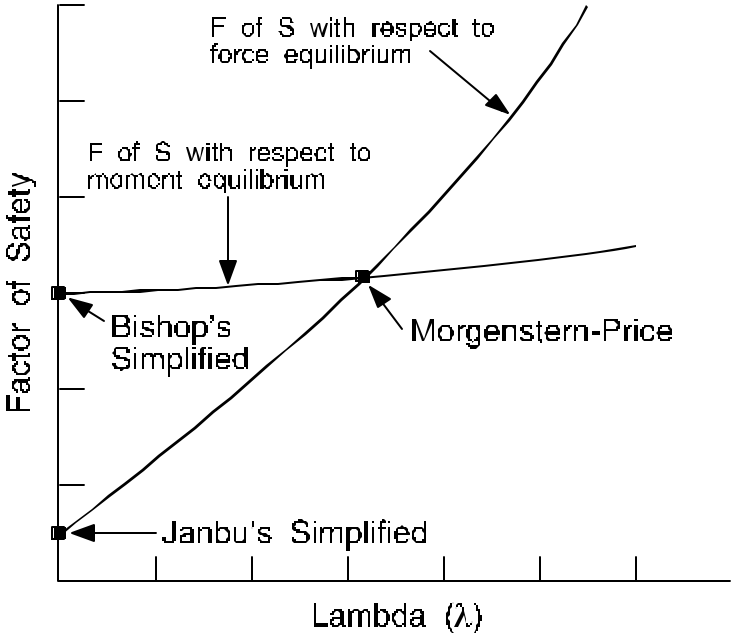
Adopting a Method

SLOPE/W can compute the factor of safety for many methods. A question often asked is, "Which method gives the best value?" There is no single answer to this question. However, there are some important factors that need to be considered when making the decision.

From a mathematical viewpoint, the best factor of safety is obtained from the methods that satisfy both force and moment equilibrium (e.g., Spencer, Morgenstern-Price and GLE). However, even with these methods, it is necessary to make some assumptions about the interslice shear forces.

The differences between all the methods and inherent assumptions can be illustrated by plotting the factor of safety versus lambda, as shown in Figure 7.1. Lambda represents the relationship between the shear and normal interslice forces. A lambda value of zero means there is no shear between the slices, and a nonzero lambda value means there is shear between the slices.

Figure 7.1 Moment and Force Factors of Safety as a Function of the Interslice Shear Force



As outlined in the Definition of Variables section of Chapter 8, SLOPE/W computes a moment factor of safety and a force factor of safety. When these two factors of safety are computed for a range of lambda values, it is possible to demonstrate the variation in factor of safety with respect to moment and force equilibrium. The two types of curves that result are shown in Figure 7.1.

Bishop's Simplified method considers normal interslice forces with no shear $(l = 0)$ and satisfies only the overall moment equilibrium. Bishop's Simplified factor of safety is therefore plotted where the moment curve intersects the vertical factor of safety axis. Janbu's Simplified method also considers normal interslice forces with no shear $(l = 0)$ but satisfies only the overall force equilibrium. Janbu's Simplified factor of safety is therefore plotted where the force equilibrium curve intersects the vertical factor of safety axis. The Spencer and Morgenstern-Price methods satisfy both moment and force equilibrium, and the factor of safety for these methods is plotted where the moment and force equilibrium curves cross at some nonzero lambda value.

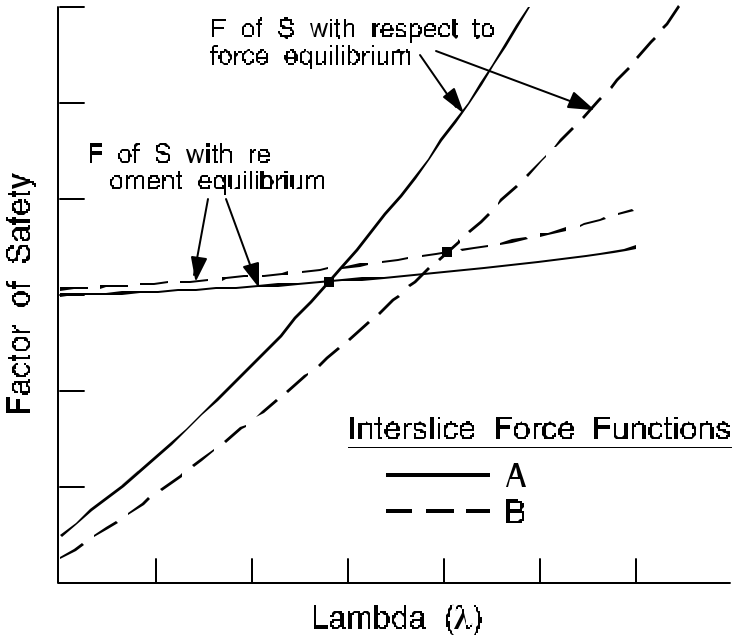
The graph in Figure 7.1 shows that the moment equilibrium curve is relatively flat, while the force equilibrium curve is relatively steep. The implication is that methods which satisfy only moment equilibrium are relatively insensitive to the assumption about the interslice shear forces. A realistic factor of safety may be obtained even though the

interslice shear force assumption is unrealistic. On the other hand, methods that satisfy only force equilibrium are relatively sensitive to the interslice shear force function.

The Bishop's Simplified factor of safety is often approximately the same as the Morgenstern-Price factor of safety. The reason for this is that moment equilibrium is insensitive to the interslice shear forces; that is, the moment equilibrium curve in a factor of safety versus lambda plot is nearly horizontal. This means that an acceptable factor of safety can sometimes be obtained from Bishop's Simplified method without the extra computations required for a Morgenstern-Price analysis.

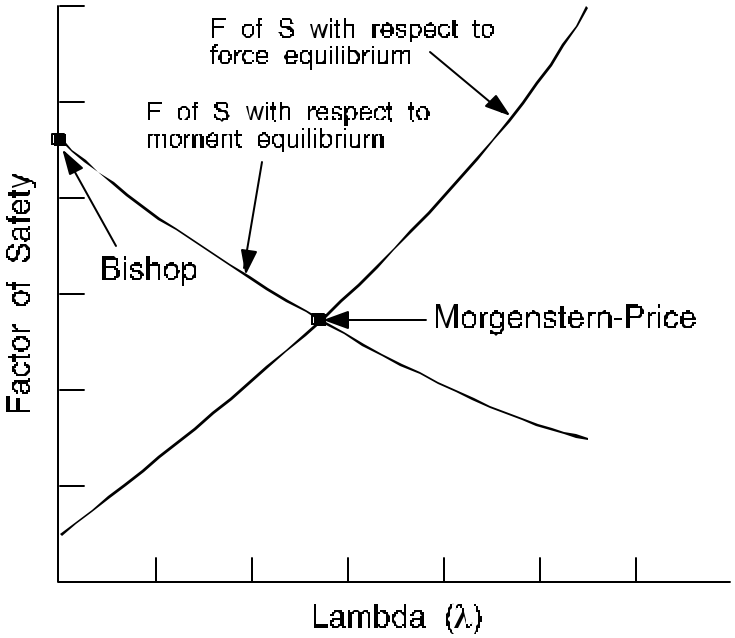
Figure 7.2 shows the effect of different interslice force functions. The interslice force function can significantly affect the force factor of safety but only has a minor effect on the moment factor of safety. This is why the Morgenstern-Price and the GLE factors of safety are often insensitive to the interslice force function. A simple constant function (Spencer method) will result in approximately the same factor of safety as a finite element based function if the moment equilibrium factor of safety versus lambda curve is nearly horizontal.

Figure 7.2 Effect of Different Interslice Force Functions



There are situations where moment equilibrium is sensitive to the interslice shear forces. In such cases, the moment equilibrium curve on a factor of safety versus lambda plot will have a significant gradient, as shown in Figure 7.3. For example, line loads and anchor loads can have a significant effect on the moment equilibrium curve.

Figure 7.3 A Case Where Moment Equilibrium Is Sensitive to the Interslice Shear Force



A factor of safety versus lambda plot helps you to decide which method is the best and what is the effect of the interslice force function. To view this plot with SLOPE/W, you can use the GLE method to specify a range of lambda values, and plot the results with the CONTOUR function.

It is good practice to carry out a GLE analysis at the start of a project to determine if the moment equilibrium is sensitive to the interslice shear forces. The results can be used as a guide in selecting the appropriate factor of safety method. For example, when moment equilibrium is not sensitive to the interslice shear force, adequate solution can be obtained using Bishop Simplified method. However, when moment equilibrium is sensitive to the interslice shear force, you may want to find the solution using the methods that satisfy both moment and force equilibrium (e.g., Spencer, Morgenstern-Price and GLE).

Local practice and experience are other factors to be considered in selecting a factor of safety method. The selected method may be less than the mathematical ideal, but it may be a suitable method provided local experience and procedures provide an acceptable understanding of the stability.

The Ordinary or Fellenius method satisfies only moment equilibrium and ignores all interslice forces. This simplification results in errors in the calculated factor of safety as high as 60 percent, (Whitman and Bailey, 1967). Consequently, the Ordinary method needs to be used with considerable caution.

Effect of Soil Properties on Critical Slip Surface

The magnitude of the strength parameters *c* and f can have an effect on the location of the slip surface with the minimum factor of safety. The two limiting cases are a purely frictional soil $(c = 0 \text{ with } f > 0)$ and a purely cohesive soil $(c > 0 \text{ with } f = 0)$.

For a purely frictional soil, the minimum factor of safety approaches the infinite slope case at an angle a, where,

$$\text{Factor of Safety} = \frac{\tan f}{\tan a}$$

The rotation center for the infinite slope case is theoretically at infinity. The consequence is that a grid of rotation centers cannot be defined such that the minimum center falls inside the grid. The minimum will always be on the edge of the grid that gives the largest radius. If you move the grid further out, the minimum will still be on the edge of the grid, because the rotation center is theoretically undefined. However, the computed factor of safety will remain relatively constant.

Furthermore, the factor of safety is independent of the slope height if *c* is specified as zero. This is usually not the case, since the slope height generally affects stability. To correctly model the effect of height, it is necessary to assign some cohesive strength to the soil. A feature of SLOPE/W that can be useful in such cases is the bilinear failure envelope. At low stresses, the material may exhibit no cohesion, but as the stress level increases due to a height increase, the material will exhibit an apparent cohesion. Applying the bilinear failure envelope may result in more realistic factors of safety. To define a bilinear failure envelope, see KeyIn Soil Properties in Chapter 4.

When a homogeneous slope is defined as a purely cohesive soil, the critical slip surface depth tends toward infinity. As with the infinite slope case, it is usually impossible to define a grid of rotation centers that contains the minimum factor of safety within the grid, since the minimum rotation center will tend to be on the edge of the grid regardless of the grid position. A further consequence is that the factor of safety tends to decrease as the radius increases.

The most realistic position of the critical slip surface is usually obtained when you use the effective strength parameters *c*' and f'.

Another useful feature of SLOPE/W that helps predict the position of the critical slip surface is the ability to model unsaturated soil behavior. In unsaturated zones, the pore-water pressure is negative. The negative pore-water pressures increase the soil strength, which is equivalent to an increase in cohesive strength. Therefore, from a modelling perspective, considering negative pore-water pressure in a stability analysis is similar to assigning the material a cohesive strength. The critical slip surface is then located at some depth, and the factor of safety is dependent on the slope height.

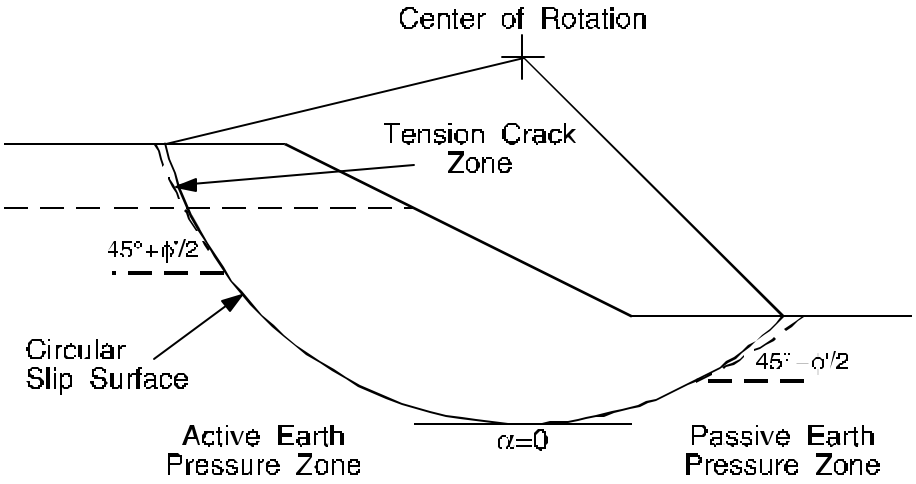
In summary, it is difficult (if not impossible) to use SLOPE/W to find a minimum factor of safety for the infinite slope case when $c = 0$ and $f > 0$ and for the undrained homogeneous case when $c > 0$ and $f = 0$.

Steep Slip Surfaces

Excessively steep slip surface segments can create convergence problems for SLOPE/W. With the Grid & Radius option, a slip surface may enter the crest area of the slope at a very steep (near-vertical) angle and exit the toe area of the slope at a steep angle. As discussed in the Unrealistic m-alpha Values section of Chapter 8, steep slip surface segments create numerical problems due to the corresponding m_a values becoming unacceptably high.

From a modelling viewpoint, the slip surface should enter the slope near the active earth pressure angle $\left(\frac{45+f'}{2}\right)$ and exit the slope near the passive earth pressure angle $\left(\frac{45-f'}{2}\right)$, as illustrated in Figure 7.4.

Figure 7.4 Active and Passive Earth Pressure Zones of a Slope



For circular or composite slip surfaces defined with the grid and radius option, convergence problems associated with m_a can be controlled by limiting the slip surface inclination. You can specify angles in both the active and passive pressure zone to limit the slip surface from being too steep. With these angles specified, the slip surface is projected out to the ground surface when the steepness of the slip surface exceeds your specified values.

Theoretically, the active and passive projection angles should be in the range as shown in Figure 7.4, but SLOPE/W allows you the flexibility to specify values in a range beyond the theoretical values. This gives you the opportunity to use your judgment in limiting the steepness of the slip surfaces.

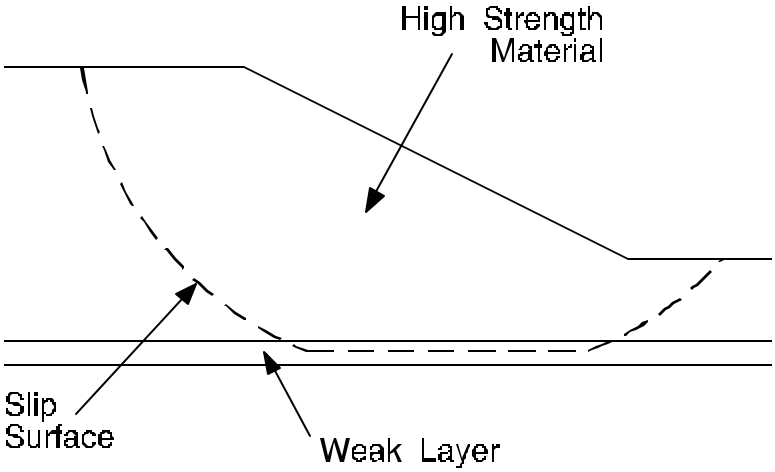
Another way of dealing with excessively steep slip surfaces in the active pressure zone is to specify a tension crack line or a tension crack angle. When you specify a tension crack line, the slip surface is projected vertically upward to the ground surface when the slip surface intersects the tension crack line. Similarly, when you specify a tension crack angle for circular or composite slip surfaces, the slip surface is projected vertically upward to the ground surface when the slip surface inclination exceeds the specified angle.

See the KeyIn Tension Crack and KeyIn Slip Surface: Grid & Radius commands in Chapter 4 for additional information on specifying tension cracks and projection angles.

Weak Subsurface Layer

Another situation which can cause numerical convergence problems occurs when a strong material overlies a very weak material, as shown in Figure 7.5.

Figure 7.5 A High Strength Material Over a Weak Layer



In the extreme case, if the weak material has essentially no strength, the upper high strength material needs to sustain tensile stresses for the slope to remain stable. If the high strength material cannot sustain tensile stresses, the slope simply collapses, and convergence problems occur.

A procedure for overcoming these numerical difficulties is to initially assign a high strength to the weak layer. Then, decrease the strength in small increments until the factor of safety is near 1.0. This process reveals the minimum strength that the weak layer can have in order to maintain stability. In this case, numerical difficulties tend to arise when the factor of safety is well below 1.0.

As a broad observation, convergence difficulties are often encountered when the model is beyond the point of limiting equilibrium or the sliding mode is physically inadmissible.

Seismic Loads

SLOPE/W simulates the effects of seismic loads due to an earthquake with pseudo-static body forces. The additional forces created by seismic or earthquake accelerations can be specified using the KeyIn Load: Seismic Load command as a vertical seismic coefficient and a horizontal seismic coefficient. SLOPE/W applies the additional forces to the centroid of each slice equal to the slice weight multiplied by the user-defined seismic coefficients.

The vertical seismic coefficient can be specified as positive or negative. A positive coefficient signifies that the vertical force is applied in the same direction as the weight of the slice, while a negative coefficient signifies the vertical force is applied in the opposite direction as the weight. Since the weight of a slice and the vertical seismic force are acting in the same line of action, SLOPE/W adds the vertical seismic force to the weight of the slice. Depending on the shear strength properties and the geometry of a slope, an increase in the weight of the slices may not always decrease the factor of safety. Therefore, you should try the analysis with both positive and negative vertical seismic coefficients in determining the critical factor of safety.

The horizontal seismic coefficient can only be specified as positive. A positive horizontal coefficient signifies that the additional force is acting horizontally in the same direction as the movement of the slope.

The magnitude of the seismic force (k*W) for each slice can be viewed with the View Slice Forces command. Figure 7.6 illustrates the free body diagram and force polygon of an example slice with a horizontal seismic coefficient of 0.25. The additional horizontal force of 86.878 units acting inside the slice is exactly equal to 25% of the weight of the slice.

Figure 7.6 Free Body Diagram and Force Polygon of a Slice Showing the Application of a Horizontal Seismic Force

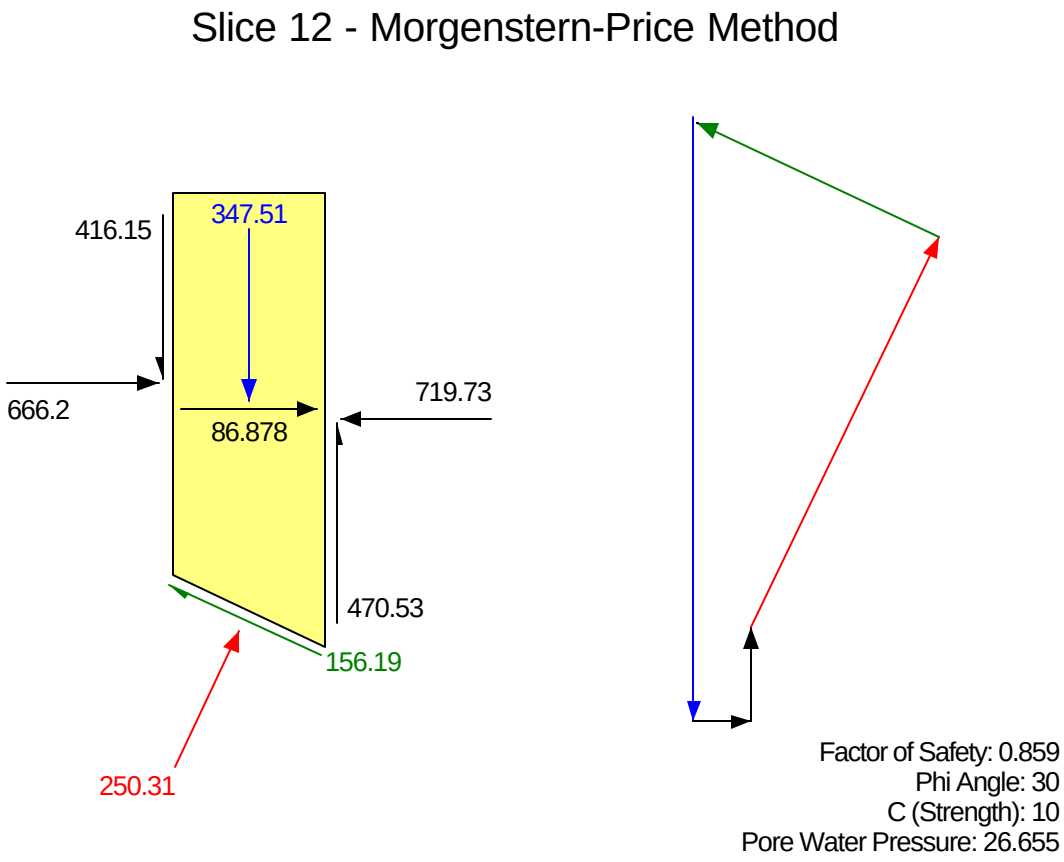


Figure 7.7 shows how the seismic force is computed for a submerged slice. Water impounded against a slope is not included in the seismic force calculation. Surface pressure loads, line loads and anchor loads are also not included in the seismic force calculation.

When you apply a seismic force, it is good practice to try a range of seismic coefficients (horizontal and vertical) and plot the factor of safety versus the seismic coefficients, as shown in Figure 7.8. The resulting relationship should show a gradual decrease in the factor of safety as the seismic coefficient values increase. Producing such a plot helps to ensure that the results are reasonable.

Figure 7.7 Application of Seismic Force for a Submerged Slice

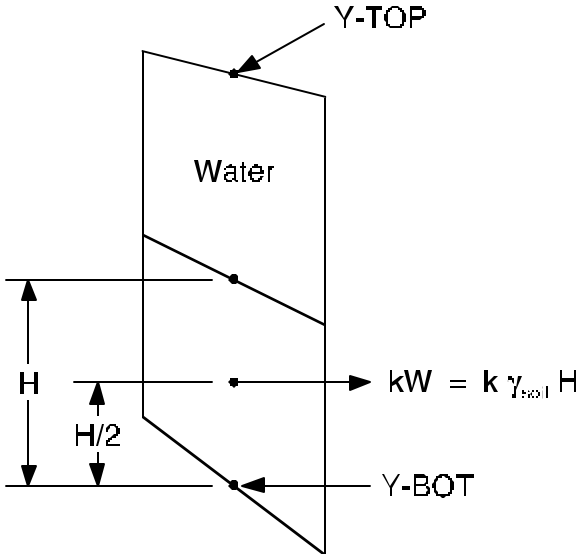
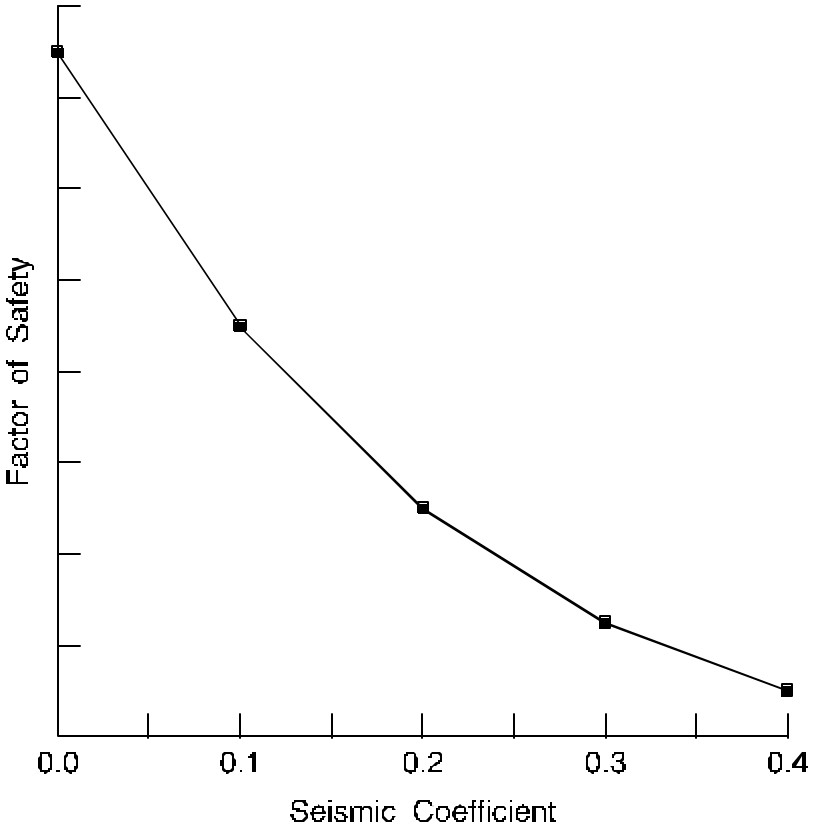


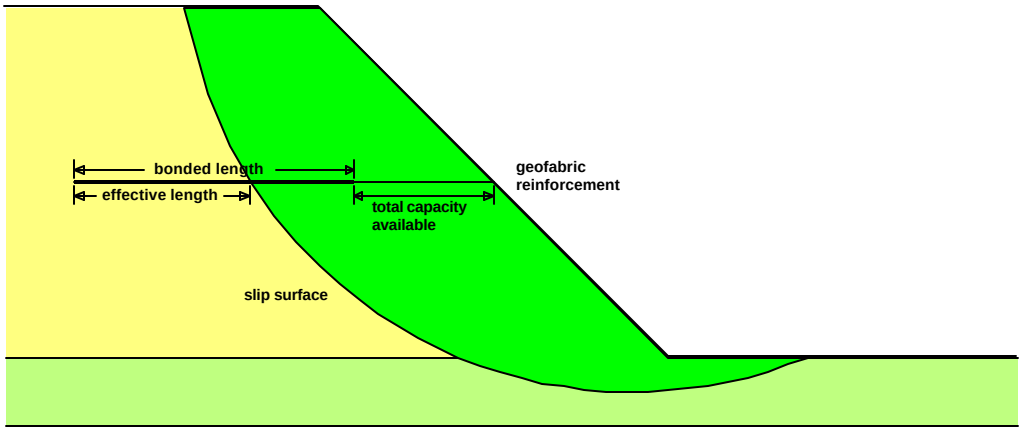
Figure 7.8 Effect of Seismic Coefficient on the Factor of Safety



Geofabric Reinforcement

A geofabric reinforcement can be modelled with the Draw Anchor Loads command. This feature makes it possible to apply a concentrated line load on the potential sliding mass arising from the fabric. Figure 7.9 illustrates a typical situation.

Figure 7.9 Modelling a Geofabric as an Anchor Load



In this case you can think of the bonded length as the embedment required to make the pull-out resistance equal to the tear strength of the fabric. The total specified capacity could be set to the tear strength, for example. Any potential slip surface passing through the fabric in front of the bond length would then be resisted by the tear strength. Slip surfaces passing through the bonded length would be resisted by a portion of the pull-out resistance.

SLOPE/W assumes that the pull-out resistance is directly proportional to the bonded part behind the slip surface if you have selected the **Variable** force option. In Figure 7.9, this is labelled as the effective length. If, for example, the slip surface passes through the mid-point of the bonded length, the pull-out resistance is taken to be half of the specified total capacity. In equation form the fabric force is calculated as:

$$\text{Geofabric Force} = \frac{\text{effective length}}{\text{bonded length}} \times \text{specified total capacity}$$

The capacity must be specified with the units of force used throughout the problem (e.g., 50 kN). The specified total capacity should not be greater than the tear strength of the fabric.

You should always examine the force SLOPE/W has used in the analysis with the View Slice Forces command. The exact force used in the calculations is given on the slice base which intersect the fabric. It is up to you as the SLOPE/W user to ensure that the embedment behind the critical slip surface is indeed adequate to provide the force used in the SLOPE/W analysis.

All of this is based on the assumption that the fabric is connected to some type of structure on the slope face which prevents surface sloughing.

Structural Elements

Structural elements (e.g., piles and shear keys) are best considered in a SLOPE/W analysis as line loads using the Draw Line Loads command. If a certain segment of a potential sliding mass is assigned the strength of concrete, for example, numerical convergence problems can result. The reason is due to the extreme contrast in strength between two slices in the sliding mass.

The gravity effect of structural elements (such as concrete retaining walls) can be included by assigning to the region the unit weight of the structural material. However, the rigidity of the structural element cannot be fully included. The rigidity of the structural element can be partially included by assigning the material a typical soil strength, but not the actual strength of the structural material.

Line loads can affect the forces on a slice. The vertical component of a line load is included in the vertical summation of forces for computing the normal at the slice base. The horizontal component of a line load also affects the base normal if interslice shear is considered in the analysis. A factor of safety sensitivity analysis consequently may be required to assess the best application point of line loads.

Active and Passive Earth Pressures

SLOPE/W can be used to compute active and passive earth pressures by defining line loads using the Draw Line Loads command. However, there are two important factors to be considered for this type of analysis. The first is that the factor of safety tends toward infinity when the lateral force is near the at-rest case, and the second is that the strength parameters *c* and f must be negative for the passive case.

Consider the diagram in Figure 7.10. The factor of safety is less than 1.0 when the lateral force is less than the active force. As the lateral force increases, the factor of safety increases. At a factor of safety of 1.0, the lateral force is equal to the active case. A further increase in the lateral force results in a further increase in the factor of safety.

As the lateral force approaches the at-rest condition, the factor of safety tends toward infinity. The reason this happens is because the gravitational driving force is balanced by the lateral force. Dividing the resisting force by zero results in a factor of safety that is undefined.

The lateral forces can be greater than the at-rest case, but the strength parameters c and f must be negative in order to obtain a solution. Setting *c* and f to negative values has the physical effect of reversing the direction of the resisting shear forces.

Once on the passive side, a further increase in the lateral force results in a further decrease in the factor of safety. The passive earth force occurs when the factor of safety is again equal to 1.0.

It is also important to realize that the position of the critical slip surface changes as the lateral force changes. Theoretically, the slip surface is at an angle of $\left(45+\phi'\right)/2$ from the horizontal for the active case and at an angle of $\left(45-\phi'\right)/2$ from the horizontal for the passive case. These values can be used as a guide for specifying the slip surfaces. For the active case, convergence difficulties can become a problem as the slip surface inclination approaches $\left(45+\phi'\right)/2$ and becomes steeper than this inclination.

When examining active and passive earth pressures, you should try a range of lateral earth forces and create a plot, such as the one shown in Figure 7.10. Without such a plot, it is difficult both to judge the validity of the results and to know whether the strength parameters should be positive or negative. Furthermore, a plot can help to explain the reason for convergence problems when the lateral earth pressure is near the at-rest case.

In active and passive earth pressure problems, the solution can be sensitive to the point of application of the applied force. Solutions are fairly stable if the application point is near the lower one-third point on the wall. Serious convergence problems can arise if the point of application is on the upper half of the wall.

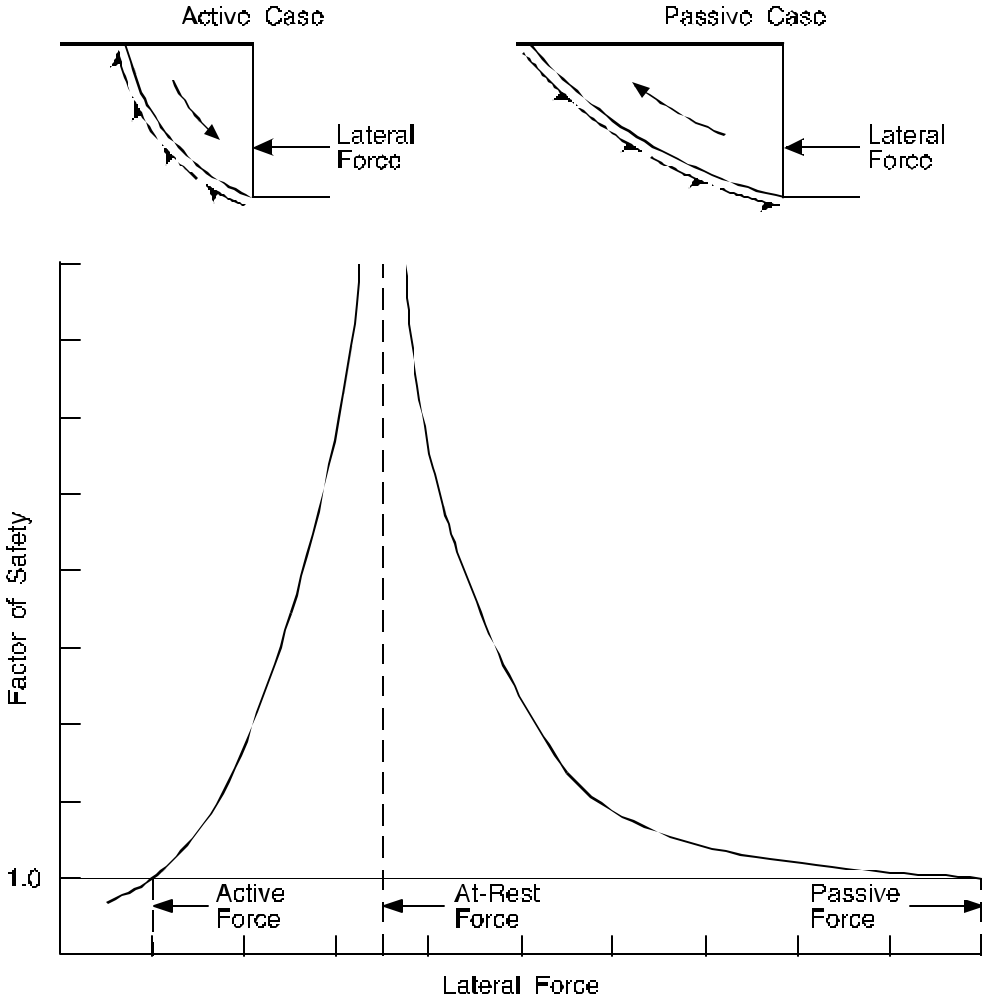
The interslice force function you select should reflect the shear on the wall. If there is no shear on the wall, then the interslice force function f(*x*) should be zero at the wall, or lambda (λ) should be zero. As described in the Interslice Forces section, the interslice shear is calculated by:

$$X = E\ l\ f(x) \tag{7.1}$$

Either lambda or f(*x*) must be zero to make the shear zero on the wall. If there is shear on the wall, then lambda multiplied by f(*x*) at the wall must be equal to the coefficient of friction; therefore, *E* times the coefficient equals *X* (the wall shear).

To obtain reliable results, you should select GLE as the analysis method (using KeyIn Analysis Method) so that you can make F of S versus Lambda (λ) plots for moment and force equilibrium. The shape and cross-over point of these two curves can be of great help in interpreting your results.

Figure 7.10 Active and Passive Earth Pressure Analysis

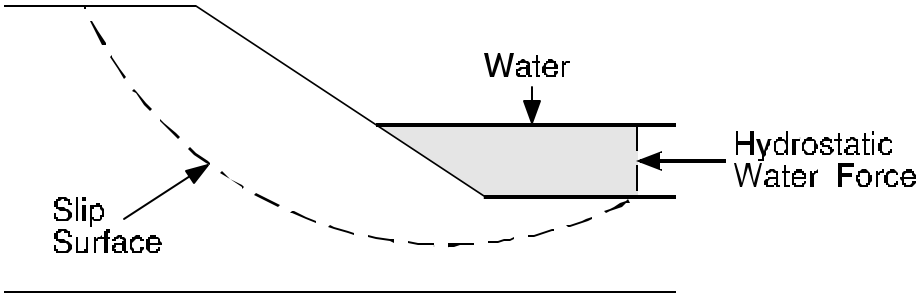


Partial Submergence

Partial submergence can be modelled in one of two ways. One is to use the Draw Pressure Lines command to apply a surface pressure to the ground surface that is representative of the fluid pressure; this procedure is described in the example with Composite Slip Surfaces in Chapter 9. The second way is to use the KeyIn Soil Properties command to model the water as a material with no strength; this procedure is described in the example with Block Slip Surfaces in Chapter 9.

When the impounded water is modelled as a material with no strength, SLOPE/W uses a vertical slip surface through the water and applies a hydrostatic horizontal force on the vertical portion of the slip surface, as illustrated in Figure 7.11.

Figure 7.11 Illustration of Partial Submergence



One factor to be aware of when modelling partial submergence with a no strength material is the interslice shear forces. Ideally, there should be no shear between the slices within the water. Practically, however, the presence of some shear between the slices does not significantly affect the factor of safety.

The interslice shear forces are not an issue for the Bishop, Janbu and Ordinary analysis methods, since these methods ignore interslice shear forces. They are only an issue when you use a method such as Spencer, Morgenstern-Price or GLE, since these methods include interslice shear forces. When using these more rigorous methods, it is best to also use an interslice side force function to reduce the shear component in the crest and toe areas. A half-sine function, for example, is better than a constant function. The analysis could be further refined with a fully specified interslice side force function.

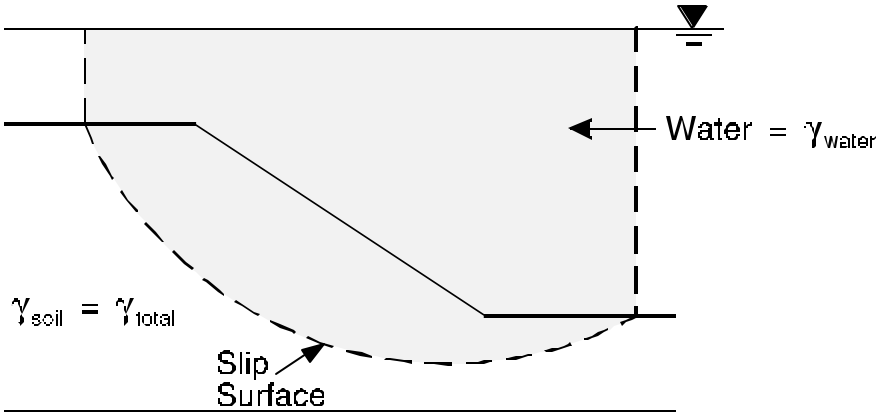
Refining the analysis with a particular side force function may not be warranted when the depth of submergence is shallow relative to the slope height. Special consideration of the interslice force function becomes more important as the depth of submergence approaches the slope height.

Complete Submergence

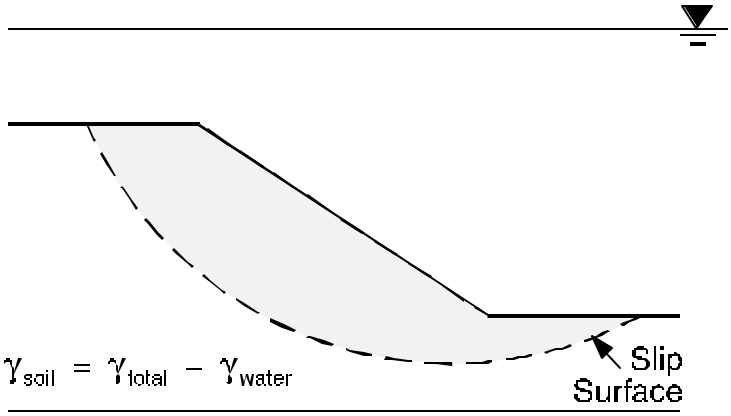
Figure 7.12a illustrates a case of complete submergence together with a potential failure mass used in the SLOPE/W analysis. While SLOPE/W can numerically handle this case, including the surface water can create some difficulties in this case. When the majority of a slice consists of water, the pore-water pressure at the slice base can become larger than the computed normal force, especially as the inclination of the slice base increases. The problem arises due to the fact that the normal is related to the slice weight multiplied by $\cos\alpha$. The $\cos\alpha$ term makes the normal less than the pore-water pressure when the slice weight (*W*) consists almost entirely of water.

Figure 7.12 Illustration of Complete Submergence

(a) Water Included in Analysis



(b) Water Not Included in Analysis



This difficulty can be avoided by not including the surface water in the analysis, as illustrated in Figure 7.12b, and by using the submerged unit weight instead of the total unit weight. The appropriate unit weight without the water is:

$$\gamma \text{ submerged} = \gamma \text{ total} - \gamma \text{ water}$$

Using the submerged unit weight is only recommended in the case of complete submergence.

Any excess pore-water pressures that exist can be defined by the usual methods; only the excess pore-water pressures must be included, and not the total pore-water pressure.

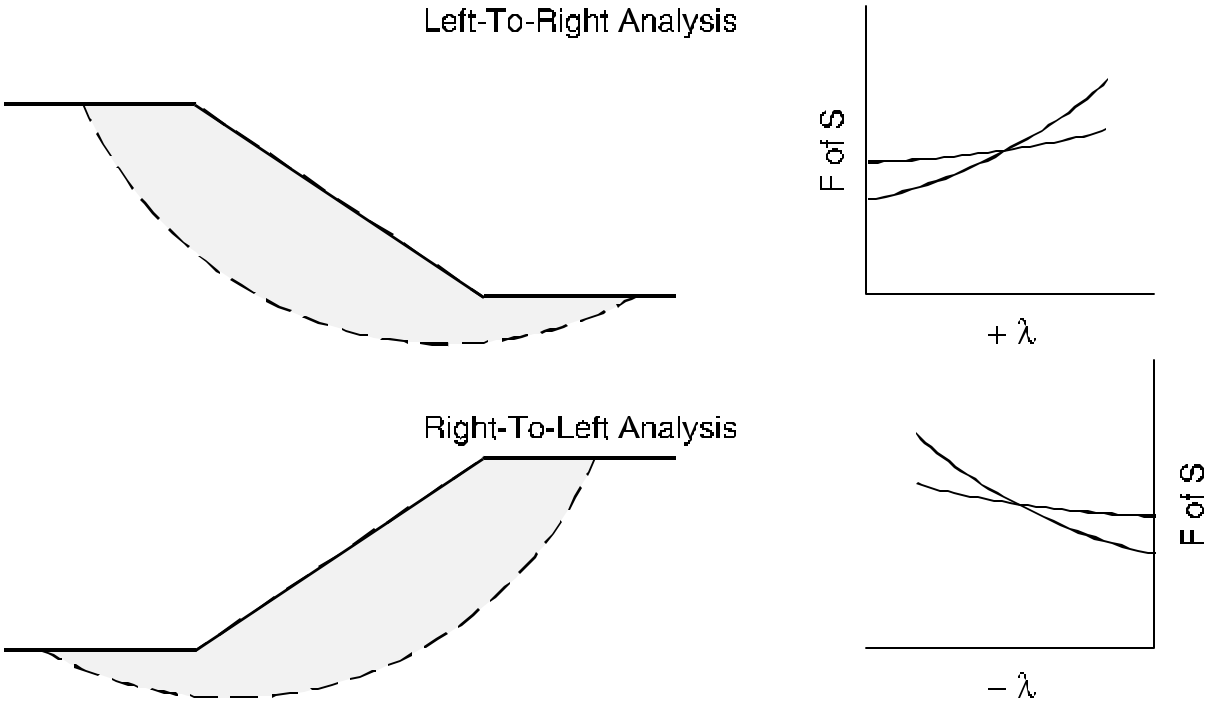
Right-To-Left Analysis

Throughout the SLOPE/W User's Guide, most cases are illustrated with a problem where the potential slide movement is from the left to the right. Cases with potential movement from the right to the left can also be accommodated by specifying this option with the KeyIn Analysis Control command. There are some issues, however, that you should be aware of in a right-to-left analysis:

- The geometry must be defined from left to right in the same way as for left-to-right problems.
- Slices are still numbered from left to right; therefore, Slice 1 is at the toe instead of the crest.
- Integration for force equilibrium begins at the toe instead of at the crest.
- Lambda is negative for methods that satisfy both force and moment equilibrium. Lambda should be specified as negative if you use the GLE method for a right-to-left problem (see Figure 7.13).

In some cases, you can expect to find small differences in factors of safety between identical left-to-right and right-to-left problems. This is because the force equilibrium starts at the crest for a left-to-right problem but starts at the toe for a right-to-left problem. The difference, however, should be insignificant.

Figure 7.13 Effect of Slope Direction on Lambda

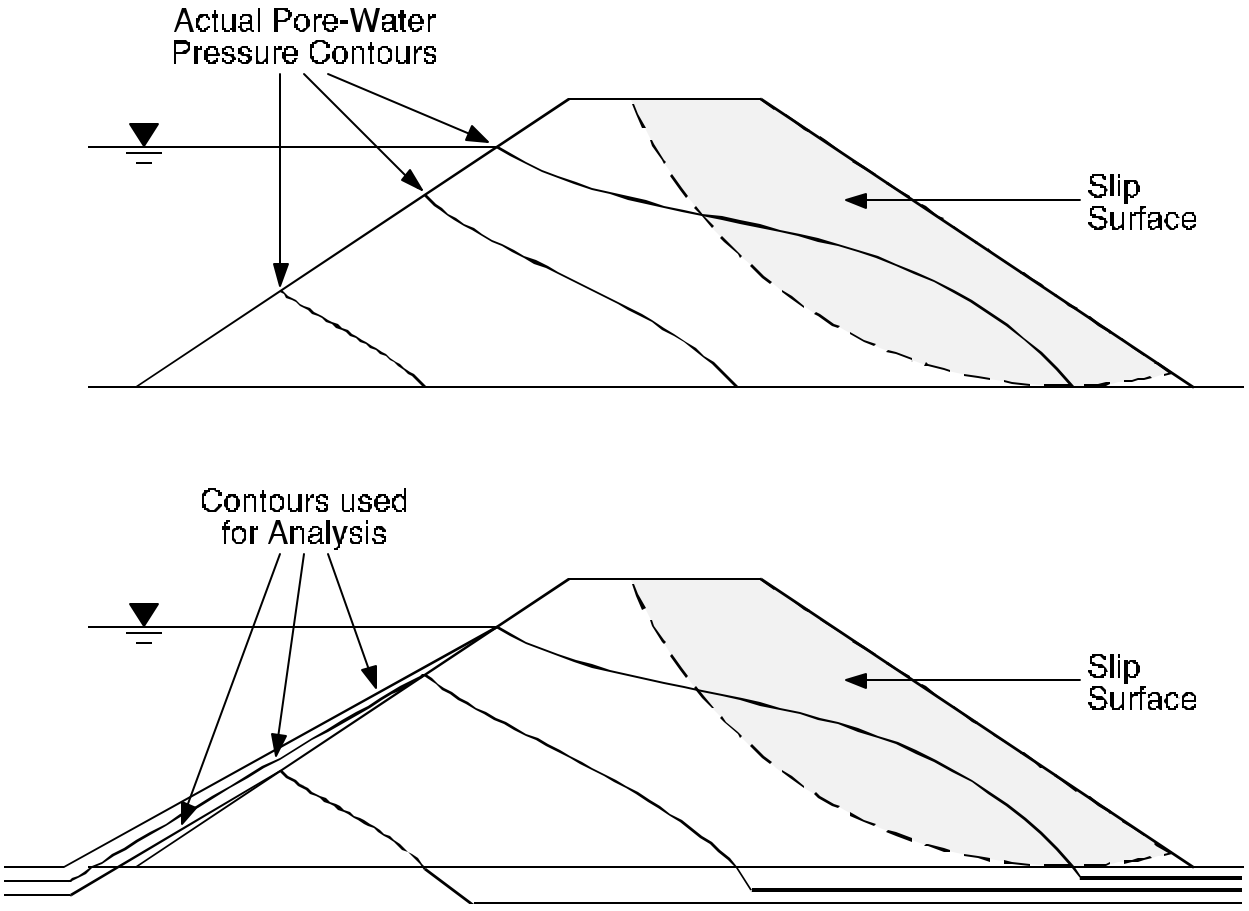


Pore-Water Pressure Contours

Figure 7.14 shows a typical stability problem where pore-water pressure contours are defined only over specific sections of the problem. In SLOPE/W, however, the contours must be defined from the left extremity to the right extremity of the problem. This can be accommodated by extending the contours below the potential failure mass, as illustrated in Figure 7.14.

This extension may not result in the actual intended pore-water pressure along the base of the problem where the contour lines are fairly steep. If the resulting approximation of pore-water pressure is considered unacceptable, you can define a grid of pore-water pressures at specific points. With a grid of data points, SLOPE/W interpolates the pore-water pressure using a more sophisticated numerical scheme (i.e., Kriging) than the simple vertical linear interpolation scheme used with the contours. See the KeyIn Analysis Control command for more information on specifying the pore-water pressure option.

Figure 7.14 Definition of Pore-Water Pressure Contours for Stability Analyses



Finite Element Stress Method

The procedure of using finite element computed stresses to compute a stability factor is a relatively new method compared to the well established method of slices based on limit equilibrium. The Finite Element Stress analysis method consequently needs to be used with considerable care and understanding. The following are some factors that you need to be aware of when computing stability factors using the Finite Element Stress method.

- SIGMA/W always output stresses as "total" stresses, together with a separate file of pore-water pressures. Consequently, if you want to do an effective stress stability analysis, you need to also specify the pore-water pressure conditions in SLOPE/W.

- Pore-water pressure conditions can be obtained from a SIGMA/W analysis or a SEEP/W analysis or they can be specified using any one of the other available options in SLOPE/W. For example, the total stresses might be obtained from a SIGMA/W stress file and the pore-water pressure conditions might be specified by a piezometric line in SLOPE/W. The main point is that the pore-water pressure conditions can be specified independent of the SIGMA/W analysis.

- The SLOPE/W problem must lie within the finite element mesh used in the SIGMA/W analysis; no part of the SLOPE/W problem can be outside the mesh. An exception to this is a no strength material, such as water. Since the base of a slice cannot exist within a no strength material, this material can be outside the mesh.

- All forces acting on a slope must be included in the SIGMA/W analysis. Line loads, anchor loads, and surface surcharge pressures, for example, must be included in the SIGMA/W analysis. You cannot exclude these forces from the stress analysis and then later add them to the SLOPE/W analysis. In other words, the state of stress along the slip surface must be complete within the SIGMA/W analysis.

- The SIGMA/W analysis must include the insitu stresses plus any change in stress that may arise due to changes in boundary conditions. The change in stresses that arise from applied surface forces alone are not sufficient in a stability analysis. The total stress must include the change in stresses and the initial insitu ground stresses.

See the KeyIn Analysis Methodl command for more information on selecting an analysis method.

Probabilistic Analysis

Probabilistic slope stability analysis can be performed with SLOPE/W Version 4. The following are important facts about doing a probabilistic analysis with SLOPE/W:

- The use of a probabilistic analysis will not affect the deterministic solution. SLOPE/W computes the factor of safety of all slip surface first and determines the critical slip surface as if no probabilistic analysis is chosen.
- A probabilistic analysis is performed on the critical slip surface only.
- The factor of safety presented on the SOLVE main window during the probabilistic analysis is the deterministic minimum factor of safety of all computed slip surfaces; however when the analysis is completed, the factor of safety presented on the SOLVE main window is the mean factor of safety of all Monte Carlo trials.
- In a probabilistic analysis, the input value of a parameter represents the mean value, and the variability of the parameter is assumed to be normally distributed with a known standard deviation.
- During the Monte Carlo trial, it is not unusual that some of the trials may not have a converged solution (i.e., factor of safety = 999.0). When this happen, the 999.0 factor of safety is not considered in the statistical analysis.
- The probabilistic results can be viewed in CONTOUR with the Draw Probability command. The results are graphically presented as the probability density function and the probability distribution function.
- The number of Monte Carlo trials is dependent on the level of confidence and the amount of variability in the input parameters. Theoretically, the greater the number of Monte Carlo trials, the more accurate the solution. It is important that you have done a sufficient number of Monte Carlo trials in a probabilistic analysis. One way to check this is to re-run the analysis with the same number of Monte Carol trials; if the two solutions are quite different, you should increase the number of Monte Carlo trials until the difference is insignificant.

If you are interested in examining the probability of failure for a slip surface other than the one with the minimum deterministic factor of safety, you should define only one slip surface and reanalyze the problem. After computing the probability of failure for the slip surface, you can move the slip surface to a different position and quickly repeat the analysis. Generally, the lowest Reliability Index and highest probability of failure occur for the slip surface with the lowest deterministic factor of safety. To increase your confidence in the results, you should consider some slip surfaces that are close to the minimum in order to confirm that the minimum factor of safety slip surface also gives the lowest reliability index.

See the KeyIn Analysis Control command for information on how to specify a probabilistic analysis. See the Probabilistic Slope Stability Analysis section in the Theory chapter for further discussion on how SLOPE/W performs a probabilistic analysis.

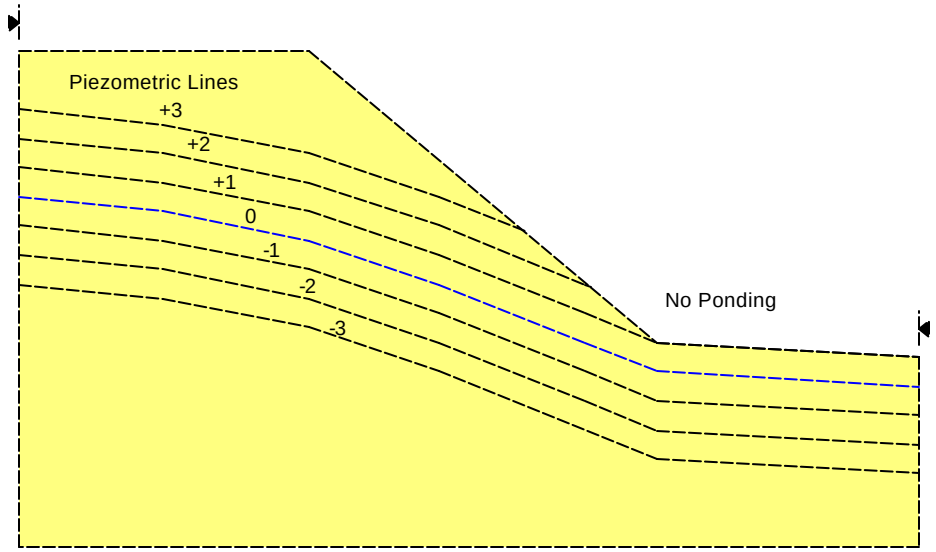
Variability in Pore-Water Pressure

In a SLOPE/W probabilistic analysis, the variability in pore-water pressure is represented by a standard deviation of the pore-water pressure head. This standard deviation is specified using the KeyIn Analysis Control command in DEFINE. In the beginning of a Monte Carlo trial, a net change in the pore-water pressure head is computed based on a random number and the normal distribution of the pore-water pressure variability. Depending on the random number, the net change can be positive and negative. The net change is then added to the pore-water pressure conditions of the entire slope.

In the case where the resulting pore-water pressure generates an equivalent water table higher than the ground surface of the slope, SLOPE/W restricts the resulting pore-water pressure head to be not higher than the ground surface. Figure 7.15 illustrates the Monte Carlo trial positions of the piezometric lines used in the analysis when the standard deviation is 1 unit for standard deviates between +3 to -3. Please note that the piezometric lines for +2 and +3 standard deviates are restricted in the downstream slope so that no ponding is allowed.

When the pore-water pressure at the base of a slice is higher than the weight of the slice, the slice has a negative effective normal which may cause convergence difficulties in solving the factor of safety equations. Restricting the piezometric line to the slope surface will limit convergence difficulties by preventing extremely high pore-water pressures at the base of the slices.

Figure 7.15 Estimation of Piezometric Lines Positions in a Monte Carlo Trial

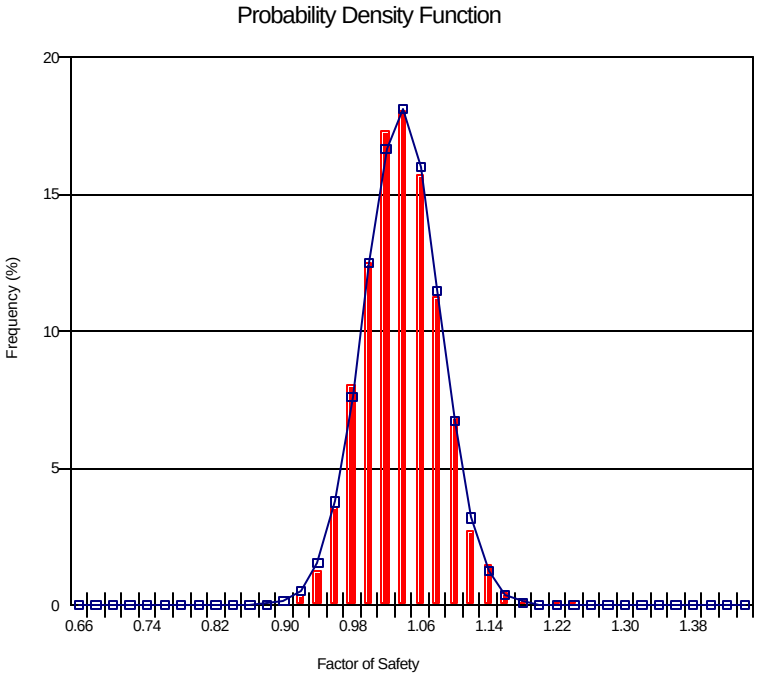


It is physically more correct to restrict the maximum rise of the pore-water pressure head to the ground surface; however, this restriction introduces some bias to the variability of the pore-water pressure conditions in the slope during the Monte Carlo trials. That is, the variability of the pore-water pressure along the downstream slope is no longer normally distributed. This bias has some effect on the probability density distribution function of the trial factors of safety.

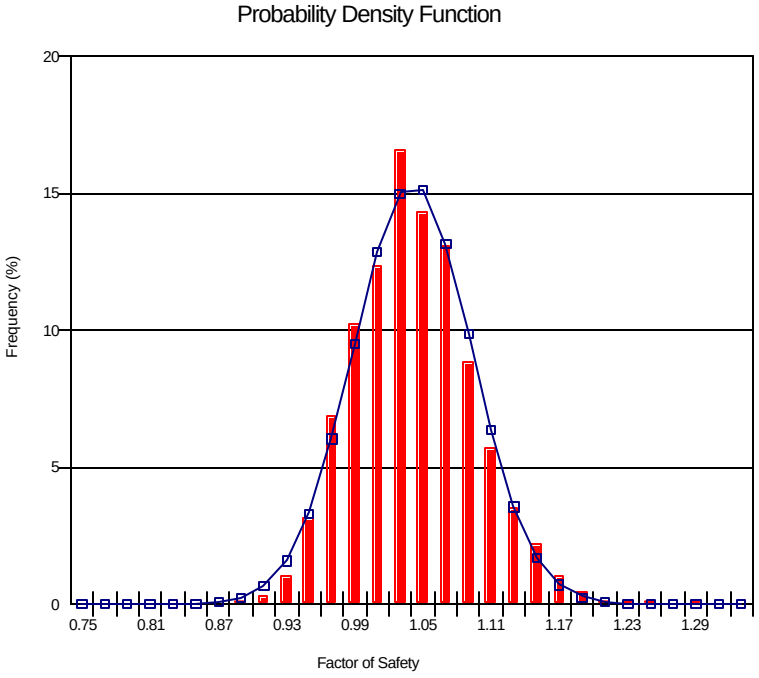
Figure 7.16 shows the probability density distribution for different degrees of pore-water pressure variability. The top diagram shows that the histogram matches nicely with the normal distribution curve when there is no variability to the pore-water pressure condition (i.e., no restriction to the trial piezometric lines). However, as the variability to the pore-water pressure condition increases, more restrictions are imposed on the trial pore-water pressure, and consequently, the histogram begins to deviate from the normal distribution curve.

Figure 7.16 Probability Density Function for Different Degrees of Pore-water Pressure Variation

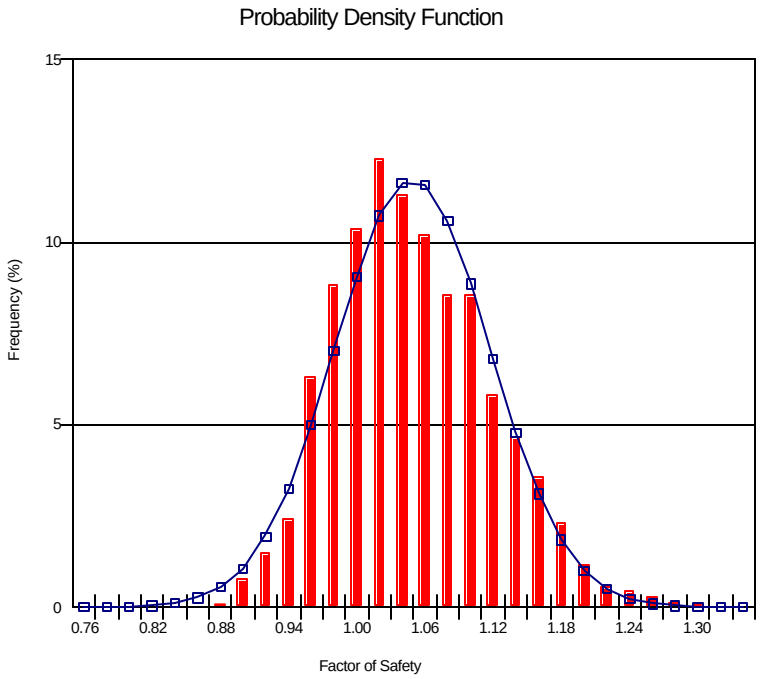
a) Standard Deviation = 0



b) Standard Deviation = 1



c) Standard Deviation = 2



Chapter 8 Theory

Introduction **8-3**

Definition Of Variables **8-4**

General Limit Equilibrium Method **8-9**

Moment Equilibrium Factor Of Safety **8-10**

Force Equilibrium Factor Of Safety **8-11**

Slice Normal Force at the Base **8-12**
Unrealistic m-alpha Values 8-13

Interslice Forces **8-16**
Corps of Engineers Interslice Force Function 8-18
Lowe-Karafiath Interslice Force Function 8-19
Fredlund-Wilson-Fan Interslice Force Function 8-20

Effect Of Negative Pore-Water Pressures **8-23**
Factor of Safety for Unsaturated Soil 8-23
Use of Unsaturated Shear Strength Parameters 8-24

Solving For The Factors Of Safety **8-25**
Stage 1 Solution 8-25
Stage 2 Solution 8-25
Stage 3 Solution 8-26
Stage 4 Solution 8-27

Simulation of the Various Methods **8-30**

Spline Interpolation of Pore-Water Pressures **8-34**

Finite Element Pore-Water Pressure **8-36**

Slice Width **8-37**

Moment Axis **8-39**

Soil Strength Models **8-41**
Anisotropic Strength 8-41
Anisotropic Strength Modifier Function 8-42
Shear/Normal Strength Function 8-42

Finite Element Stress Method **8-44**
Stability Factor 8-44
Normal Stress and Mobilized Shear Stress 8-45

Probabilistic Slope Stability Analysis **8-48**
Monte Carlo Method 8-48
Parameter Variability 8-48
Normal Distribution Function 8-49
Random Number Generation 8-49

Estimation of Input Parameters....8-50
Correlation Coefficient....8-50
Statistical Analysis....8-51
Probability of Failure and Reliability Index....8-53
Number of Monte Carlo Trials....8-54

Introduction

This chapter explains the theory used in the development of SLOPE/W. The variables used are first defined, followed by a brief description of the General Limit Equilibrium method (GLE). The relevant equations are derived, including the base normal force equation and the factor of safety equations. This is followed by a section describing the iterative procedure adopted in solving the nonlinear factor of safety equations. Attention is then given to aspects of the theory related to soils with negative pore-water pressures.

SLOPE/W solves two factor of safety equations; one satisfying force equilibrium and one satisfying moment equilibrium. All the commonly used methods of slices can be visualized as special cases of the General Limit Equilibrium (GLE) solution.

The theory of the Finite Element Stress method is presented as an alternative to the limit equilibrium stability analysis. This method computes the stability factor of a slope based on the stress state in the soil obtained from a finite element stress analysis. Finally, the theory of probabilistic slope stability using the Monte Carlo method is also presented.

Definition Of Variables

SLOPE/W uses the theory of limit equilibrium of forces and moments to compute the factor of safety against failure. The General Limit Equilibrium (GLE) theory is presented and used as the context for relating the factors of safety for all commonly used methods of slices.

A factor of safety is defined as that factor by which the shear strength of the soil must be reduced in order to bring the mass of soil into a state of limiting equilibrium along a selected slip surface.

For an effective stress analysis, the shear strength is defined as:

$$s = c' + \left(s_n - u\right)\tan f' \tag{8.1}$$

where:

s = shear strength
c' = effective cohesion
f' = effective angle of internal friction
s_n = total normal stress
u = pore-water pressure

For a total stress analysis, the strength parameters are defined in terms of total stresses and pore-water pressures are not required.

The stability analysis involves passing a slip surface through the earth mass and dividing the inscribed portion into vertical slices. The slip surface may be circular, composite (i.e., combination of circular and linear portions) or consist of any shape defined by a series of straight lines (i.e., fully specified slip surface).

The limit equilibrium formulation assumes that:

1. The soil behaves as a Mohr-Coulomb material.

2. The factor of safety of the cohesive component of strength and the frictional component of strength are equal for all soils involved.

3. The factor of safety is the same for all slices.

Figures 8.1 and 8.2 show all the forces acting on a circular and a composite slip surface. The variables are defined as follows:

W = the total weight of a slice of width b and height h
N = the total normal force on the base of the slice
S = the shear force mobilized on the base of each slice.
E = the horizontal interslice normal forces. Subscripts L and R designate the left and right sides of the slice, respectively.
X = the vertical interslice shear forces. Subscripts L and R define the left and right sides of the slice, respectively.
D = an external line load.
kW = the horizontal seismic load applied through the centroid of each slice.
R = the radius for a circular slip surface or the moment arm associated with the mobilized shear force, S_m for any shape of slip surface.
f = the perpendicular offset of the normal force from the center of rotation or from the center of moments. It is assumed that f distances on the right side of the center of rotation of a negative

slope (i.e., a right-facing slope) are negative and those on the left side of the center of rotation are positive. For positive slopes, the sign convention is reversed.

x = the horizontal distance from the centerline of each slice to the center of rotation or to the center of moments.

e = the vertical distance from the centroid of each slice to the center of rotation or to the center of moments.

d = the perpendicular distance from a line load to the center of rotation or to the center of moments.

h = the vertical distance from the center of the base of each slice to the uppermost line in the geometry (i.e., generally ground surface).

a = the perpendicular distance from the resultant external water force to the center of rotation or to the center of moments. The L and R subscripts designate the left and right sides of the slope, respectively.

A = the resultant external water forces. The L and R subscripts designate the left and right sides of the slope, respectively.

w = the angle of the line load from the horizontal. This angle is measured counter-clockwise from the positive x-axis.

a = the angle between the tangent to the center of the base of each slice and the horizontal. The sign convention is as follows. When the angle slopes in the same direction as the overall slope of the geometry, α is positive, and vice versa.

Figure 8.1 Forces Acting on a Slice Through a Sliding Mass with a Circular Slip Surface

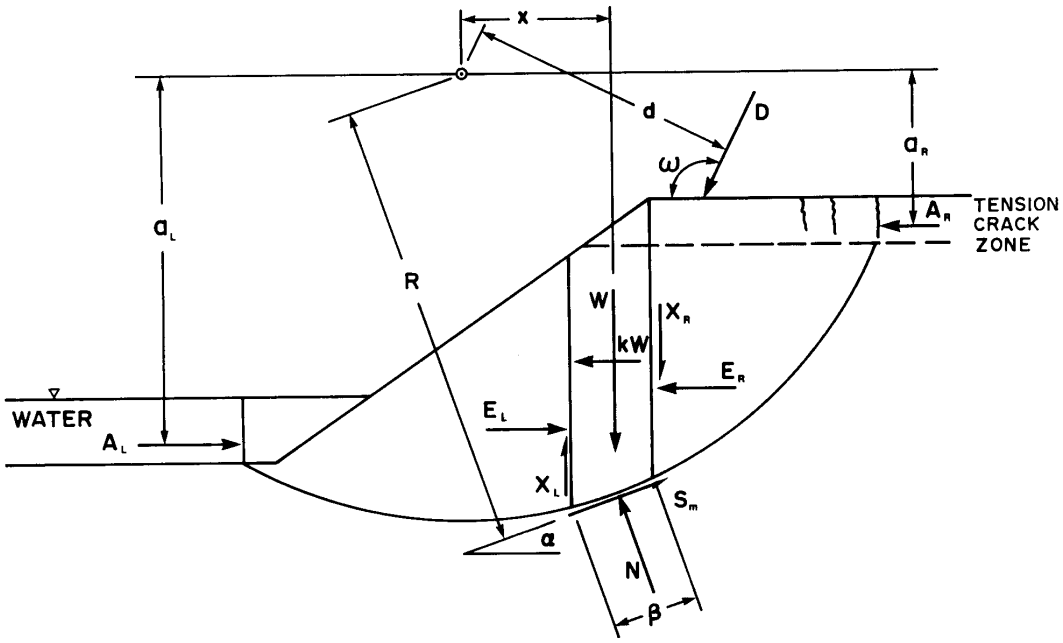


Figure 8.2 Forces Acting on a Slice through a Sliding Mass with a Composite Slip Surface

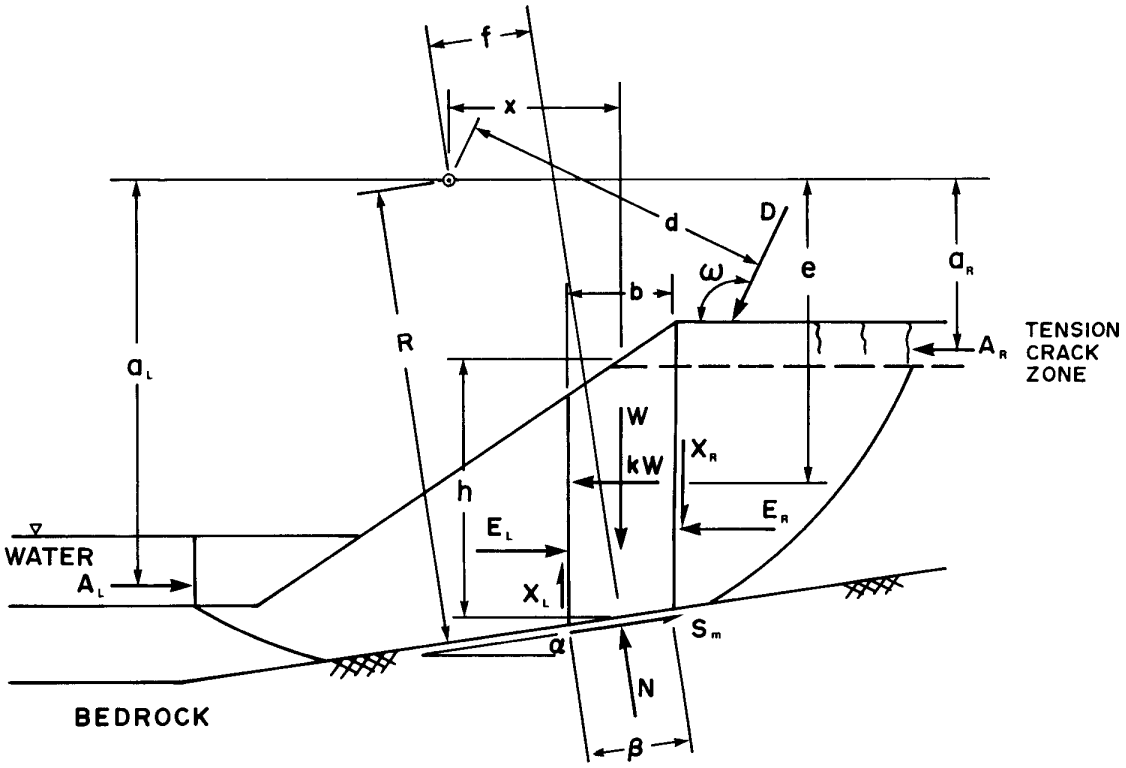
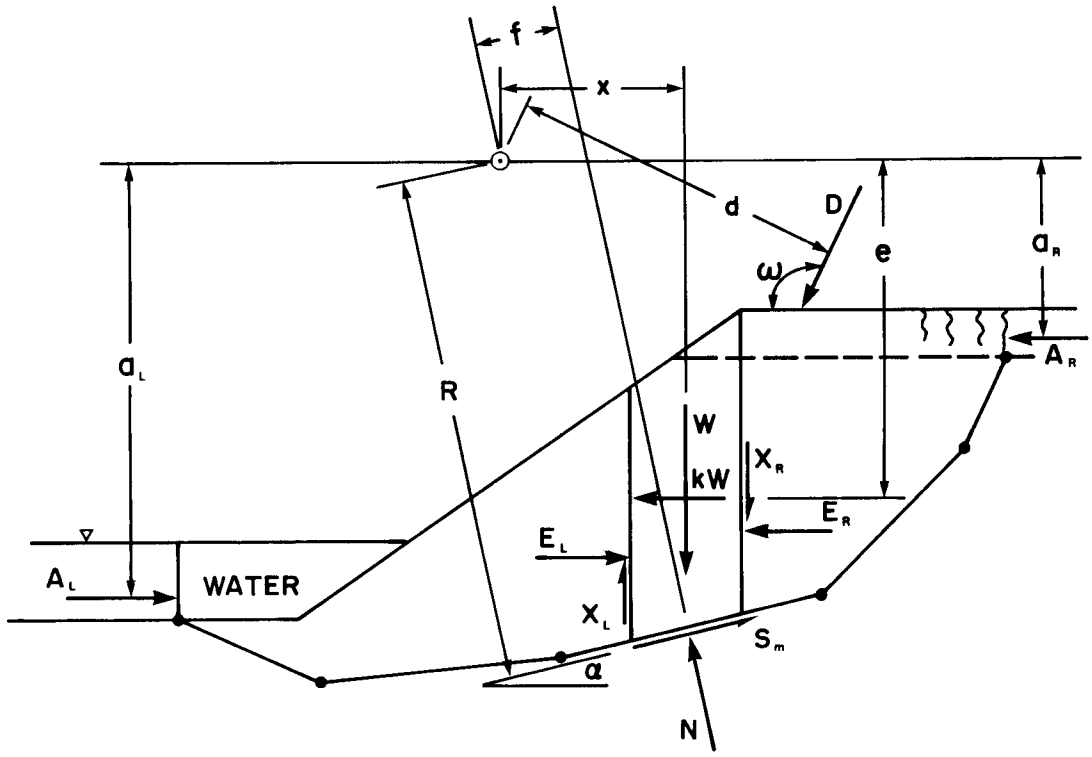


Figure 8.3 shows the forces acting on a slip surface defined by a series of straight lines. The center for moment equilibrium is immaterial when both moment and force equilibrium are satisfied. However, when only moment equilibrium is satisfied, it is important to select a reasonable center for moment equilibrium. For fully specified and block specified slip surfaces, you must define the center for moment equilibrium as the axis point.

Figure 8.3 Forces Acting on a Slice through a Sliding Mass Defined by a Fully Specified Slip Surface



The magnitude of the shear force mobilized to satisfy conditions of limiting equilibrium is,

$$S_m = \frac{s\,b}{F} = \frac{b\left(c' + \left(s_n - u\right)\tan f'\right)}{F} \tag{8.2}$$

where:

$s_n = \frac{N}{b}$ = average normal stress at the base of each slice

F = the factor of safety

b = the base length of each slice

The elements of statics that can be used to derive the factor of safety are the summations of forces in two directions and the summation of moments. These, along with failure criteria, are insufficient to make the problem determinate. More information must be known about either the normal force distribution at the base of the slices or the interslice force distribution. Tables 8.1 and 8.2 summarize the known and unknown quantities associated with a slope stability analysis.

Table 8.1 Summary of Known Quantities in Solving for the Factor of Safety

Number of Known Quantities	Description
n	Summation of forces in the horizontal direction
n	Summation of forces in the vertical direction
n	Summation of moments
n	Mohr-Coulomb Failure Criterion
4n	Total number of equations

Table 8.2 Summary of Unknown Quantities in Solving for the Factor of Safety

Number of Unknown Quantities	Description
n	Magnitude of the normal force at the base of a slice, *N*
n	Point of application of the normal force at the base of each slice
n - 1	Magnitude of the normal force at the interface between slices, *E*
n - 1	Point of application of the normal force at the interface between slices, *X*
n - 1	Magnitude of the shear force at the interface between slices, *X*
n	Shear force on the base of each slice, S_m
1	Factor of safety, *F*
1	Value of Lambda, l
6n - 1	Total number of unknowns

Since the number of unknown quantities exceeds the number of known quantities, the problem is indeterminate. Assumptions regarding the directions, magnitude, and/or point of application of some of the forces must be made to render the analysis determinate. Most methods first assume that the point of application of the normal force at the base of a slice acts through the centerline of the slice. Then an assumption is most commonly made concerning the magnitude, direction, or point of application of the interslice forces. In general, the various methods of slices can be classified in terms of (1) the statics used in deriving the factor of safety equation and (2) the interslice force assumption used to render the problem determinate.

General Limit Equilibrium Method

The General Limit Equilibrium method (GLE) uses the following equations of statics in solving for the factor of safety:

1. The summation of forces in a vertical direction for each slice. The equation is solved for the normal force at the base of the slice, *N*.

2. The summation of forces in a horizontal direction for each slice is used to compute the interslice normal force, *E*. This equation is applied in an integration manner across the sliding mass (i.e., from left to right).

3. The summation of moments about a common point for all slices. The equation can be rearranged and solved for the moment equilibrium factor of safety, F_m.

4. The summation of forces in a horizontal direction for all slices, giving rise to a force equilibrium factor of safety, F_f.

The analysis is still indeterminate, and a further assumption is made regarding the direction of the resultant interslice forces. The direction is assumed to be described by a interslice force function. The factors of safety can now be computed based on moment equilibrium $\left(F_m\right)$ and force equilibrium $\left(F_f\right)$. These factors of safety may vary depending on the percentage $\left(l\right)$ of the force function used in the computation.

The factor of safety satisfying both moment and force equilibrium is considered to be the converged factor of safety of the GLE method.

Using the same GLE approach, it is also possible to specify a variety of interslice force conditions and satisfy only the moment or force equilibrium conditions. The section entitled SLOPE/W Equations in this chapter explains how the factors of safety computed from the GLE method can be related to other commonly used methods of slices.

Moment Equilibrium Factor Of Safety

Reference can be made to Figures 8.1, 8.2, or 8.3 for deriving the moment equilibrium factor of safety equation. In each case, the summation of moments for all slices about a common point, can be written as follows:

$$\sum Wx - \sum S_m R - \sum Nf + \sum kWe \pm [Dd] \pm Aa = 0 \quad (8.3)$$

The brackets [ ] in Equation 8.3 mean that these forces are considered only for the slice on which the forces act. Substituting Equation 8.2 into Equation 8.3 and solving for the factor of safety gives,

$$F_m = \frac{\sum \left(c' bR + \left(N - ub\right) R \tan f'\right)}{\sum Wx - \sum Nf + \sum kWe \pm [Dd] \pm Aa} \quad (8.4)$$

Equation 8.4 is nonlinear since the normal force, *N*, is also a function of the factor of safety. The procedure for solving the equation is described in SLOPE/W Equations in this chapter.

Force Equilibrium Factor Of Safety

Reference can be made to Figures 8.1, 8.2, or 8.3 for deriving the force equilibrium factor of safety equation. The summation of forces in a horizontal direction for all slices gives,

$$\sum\left(E_L - E_R\right) - \sum\left(N \sin a\right) + \sum\left(S_m \cos a\right) - \sum\left(kW\right) + \left[D \cos w\right] \pm A = 0 \tag{8.5}$$

The term $\sum\left(E_L - E_R\right)$ must be zero when summed over the entire sliding mass. Substituting Equation 8.2 into Equation 8.5 and solving for the factor of safety gives,

$$F_f = \frac{\sum\left(c'b \cos a + \left(N - ub\right) \tan f' \cos a\right)}{\sum N \sin a + \sum kW - \left[D \cos w\right] \pm A} \tag{8.6}$$

Equation 8.6 is also nonlinear, and the procedure for solving the equation is described in Effect Of Negative Pore-Water Pressures in this chapter.

Slice Normal Force at the Base

The normal force at the base of a slice is derived from the summation of forces in a vertical direction on each slice.

$$-W+\left(X_L-X_R\right)+N\cos a+S_m\sin a-[D\sin w]=0 \tag{8.7}$$

Substituting Equation 8.2 into 8.7 and solving for the normal force, *N*, gives,

$$N = \frac{W + (X_R - X_L) - \frac{c'\beta \sin\alpha + u\beta \sin\alpha \tan\phi'}{F} + \left[D \sin\omega\right]}{\cos\alpha + \frac{\sin\alpha \tan\phi'}{F}} \tag{8.8}$$

The denominator in Equation 8.8 is commonly given the variable name, m_a. The factor of safety, *F*, is equal to the moment equilibrium factor of safety, F_m, when solving for moment equilibrium, and equal to the force factor of safety, F_f, when solving for force equilibrium.

Equation 8.8 cannot be solved directly since the factor of safety (*F*) and the interslice shear forces, (i.e., X_L and X_R) are unknown. The normal at the base of each slice is solved using an interactive scheme.

To commence the solution for the factor of safety, it is possible to neglect the interslice shear and normal forces on each slice (Fellenius, 1936). When forces are summed in a direction perpendicular to the base of each slice, the following equation is obtained for the normal force.

$$N=W\cos a-kW\sin a+\left[D\cos\left(w+a-90\right)\right] \tag{8.9}$$

Using the simplified equation (Equation 8.9) in solving Equations 8.4 and 8.6 provides starting values for the factor of safety computations. The factor of safety from Equation 8.4 is the Fellenius or Ordinary method factor of safety.

Next, assuming that the interslice shear forces in Equation 8.8 are equal to zero, the normal force at the base can be computed by:

$$N = \frac{W - \frac{c'\beta \sin\alpha + u\beta \sin\alpha \tan\phi'}{F} + \left[D \sin\omega\right]}{\cos\alpha + \frac{\sin\alpha \tan\phi'}{F}} \tag{8.10}$$

When Equation 8.10 is used in solving for the moment equilibrium factor of safety (i.e., Equation 8.4). The solution is the factor of safety for Bishop's Simplified method.

Equation 8.10 can be used in solving for the force equilibrium factor of safety (i.e., Equation 8.6). The solution is Janbu's Simplified method without the empirical correction factor, f_o, applied (Janbu, Bjerrum, and Kjaernsli, 1956).

If both the moment and force equilibrium equations are to be solved simultaneously, it is necessary to first compute the interslice normal forces.

Unrealistic m-alpha Values

The normal force at the base of a slice sometimes becomes unreasonable due to the unrealistic values computed for m_a represented by the denominator in Equation 8.8 and 8.10. As shown in Figure 8.4, the variable m_a is a function

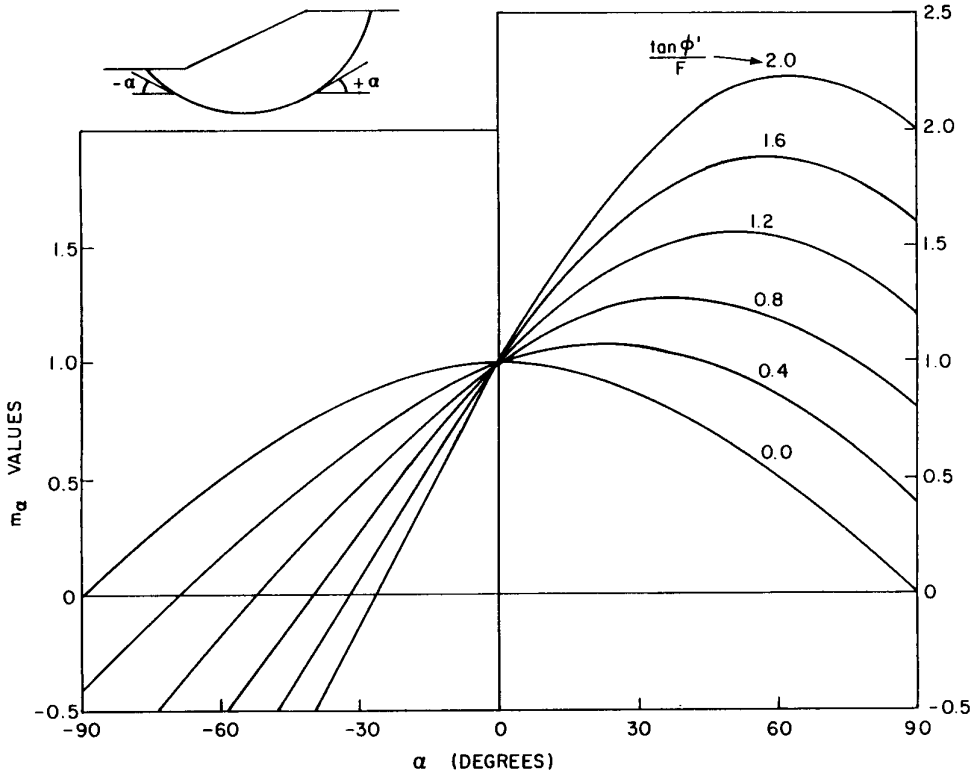
of inclination of the base of a slice, α, and $\frac{\tan f'}{F}$. Computational difficulties occur when m_a approaches zero. This situation can occur when α is negative and $\frac{\tan f'}{F}$ is large or when α is large and $\frac{\tan f'}{F}$ is small. Specifically, the m_a value will become zero when the base inclination of any slice, α, bears the following relationship to the mobilized friction angle, $\frac{\tan f'}{F}$:

$$\frac{\cos a}{\sin a} = \frac{1}{\tan a} = -\frac{\tan f'}{F} \tag{8.11}$$

When the m_a value approaches zero, the computed normal force, *N*, on the slice becomes excessively large. As a result, the mobilized shearing resistance, S_m, becomes very large and exerts a disproportionately large influence on the computation of the factor of safety.

The factor of safety calculation can take on another extreme when m_a is negative. The m_a term can be negative when the base angle of the slice, α, is more negative than the limiting angle, α_1. In this case, the computed normal force is negative. Consequently, the computed factor of safety may be under-estimated, since the total mobilized shearing resistance is reduced. When a slice has a small but negative m_a value, its normal force becomes large and negative when compared with other slices. The large, negative value then dominates the stability calculations, and the computed factor of safety can go less than zero, which of course is meaningless.

Figure 8.4 Magnitude of m_a for various α, f, and F values



Problems associated with the magnitude of m_a are mainly the result of an inappropriately assumed shape for the slip surface. Ideally, the classic earth pressure theory should be used to establish the limiting conditions for the shape of the slip surface. In applying the earth pressure theory, the soil is divided into two regions, namely an active earth pressure zone and a passive earth pressure zone (Figure 8.5). The inclination of the slip surface in the passive zone of the sliding mass should be limited to the maximum obliquity for the passive state. That is,

$$\alpha_1 < \frac{\phi'}{2} - 45 \tag{8.12}$$

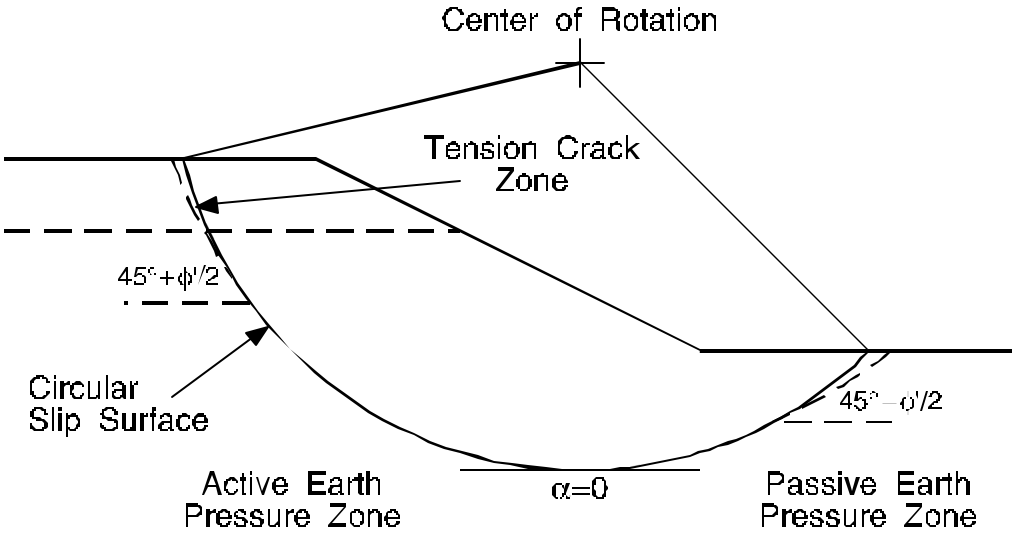
Likewise, the inclination of the slip surface in the active zone should not exceed the value obtained from the following equation:

$$\alpha_1 < \frac{\phi'}{2} + 45 \tag{8.13}$$

These solutions will generally resolve the m_a problems. The active zone also may be combined with a vertical tension crack zone to alleviate m_a problems.

It is the responsibility of the user to ensure that the limiting angles with respect to the active and passive zones are not violated. However, if the conditions are violated, there is a check in SLOPE/W to prevent the absolute value of m_a from going below 1.0×10^{-5}.

Figure 8.5 Active and Passive Earth Pressure Zones of a Slope



The inclination of the slip surface can be limited by user specified values. A value can be specified for the active pressure zone and another value can be specified for the passive pressure zone.

Interslice Forces

Interslice shear forces are required to calculate the normal force at the base of each slice. The interslice shear force is computed as a percentage of the interslice normal force according to the following empirical equation, (Morgenstern and Price, 1965):

$$X = E\ \lambda\ f(x) \tag{8.14}$$

where:

λ = the percentage (in decimal form) of the function used

$f(x)$ = interslice force function representing the relative direction of the resultant interslice force

Figure 8.6 shows some typical function shapes. The type of force function used in calculating the factor of safety is the prerogative of the user.

Figure 8.6 Functional Variation of the Direction of the Interslice Forces with Respect to the X-Direction

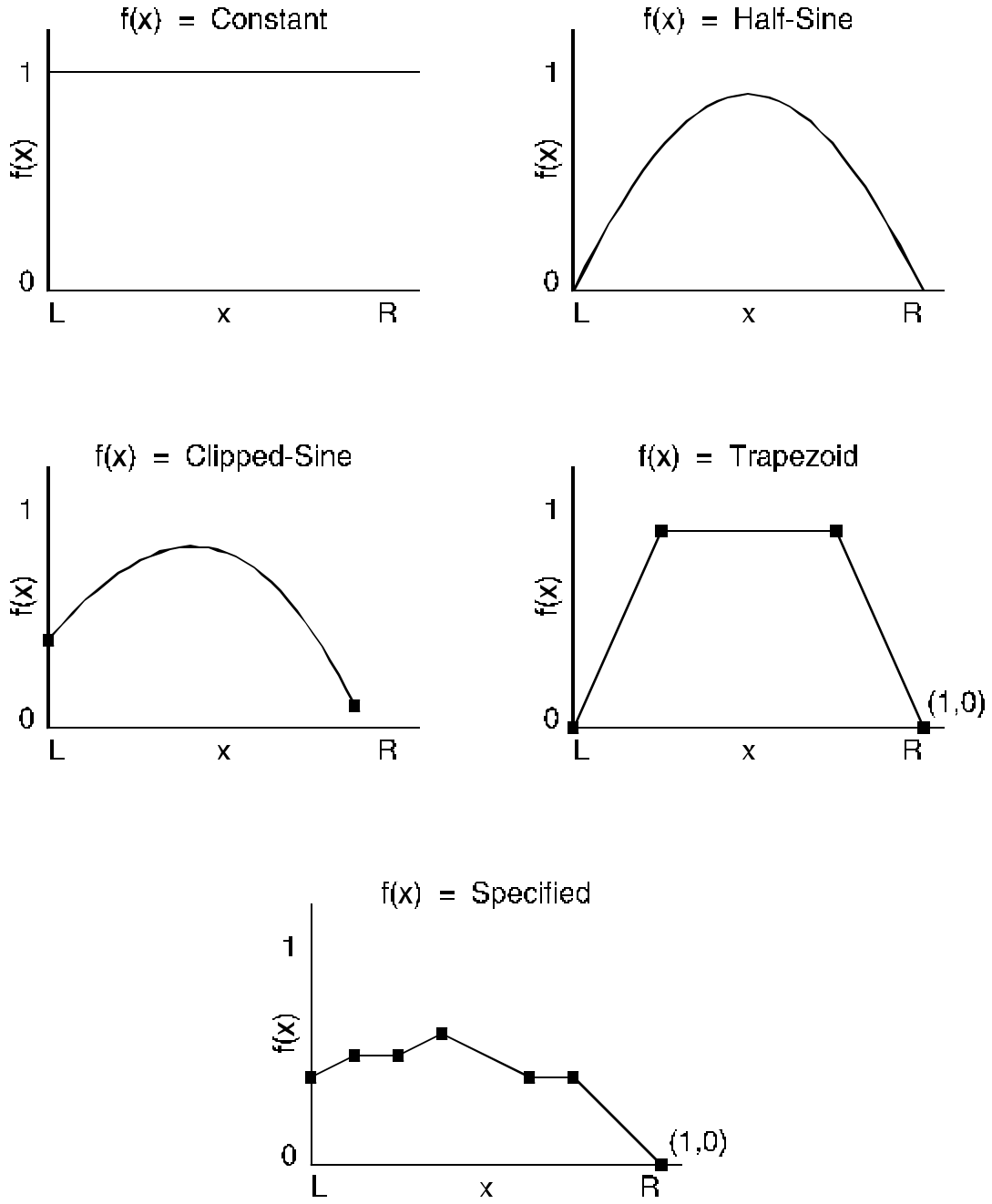


Figure 8.7 illustrates how the interslice force function $f(x)$ is used to compute the interslice shear force. Consider the use of a half-sine force function. Assume that the normal force E between Slice 1 and 2 is 100 kN, and that the applied Lambda value l is 0.5. The slice boundary is at the quarter-point along the slip surface. The $f(x)$ value at this point is 0.707 (sin 45). The shear force X then is,

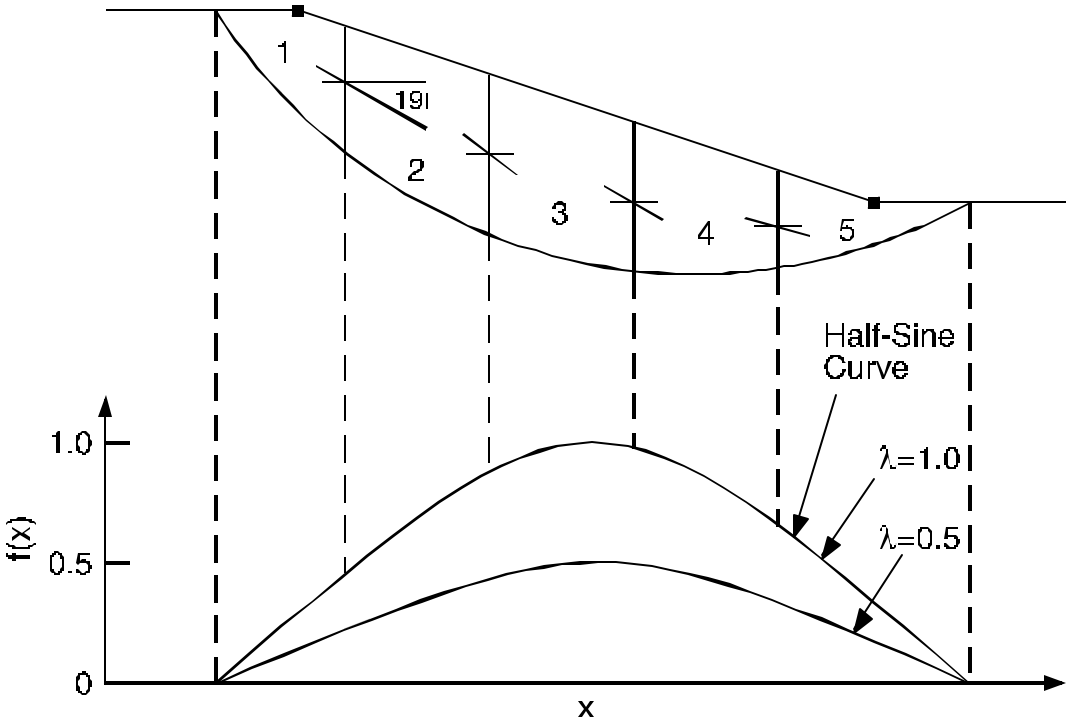
$f(x)$ = sin 45 = 0.707
l = 0.5
E = 100
X = 100 × 0.5 × 0.707 = 35.35 kN

For this example, the ratio of shear to normal varies from 0.0 at the crest and at the toe, to a maximum of 0.5 at the midpoint along the slip surface.

The summation of forces in a horizontal direction can be written for each slice.

$$(E_L - E_R) - N\sin\alpha + S_m\cos\alpha - kW + [D\cos\omega] = 0 \tag{8.15}$$

Figure 8.7 Interslice Force Convention for the General Limit Equilibrium Method (GLE)



Substituting Equation 8.2 into Equation 8.15 and solving for the interslice normal on the right side of each slice gives,

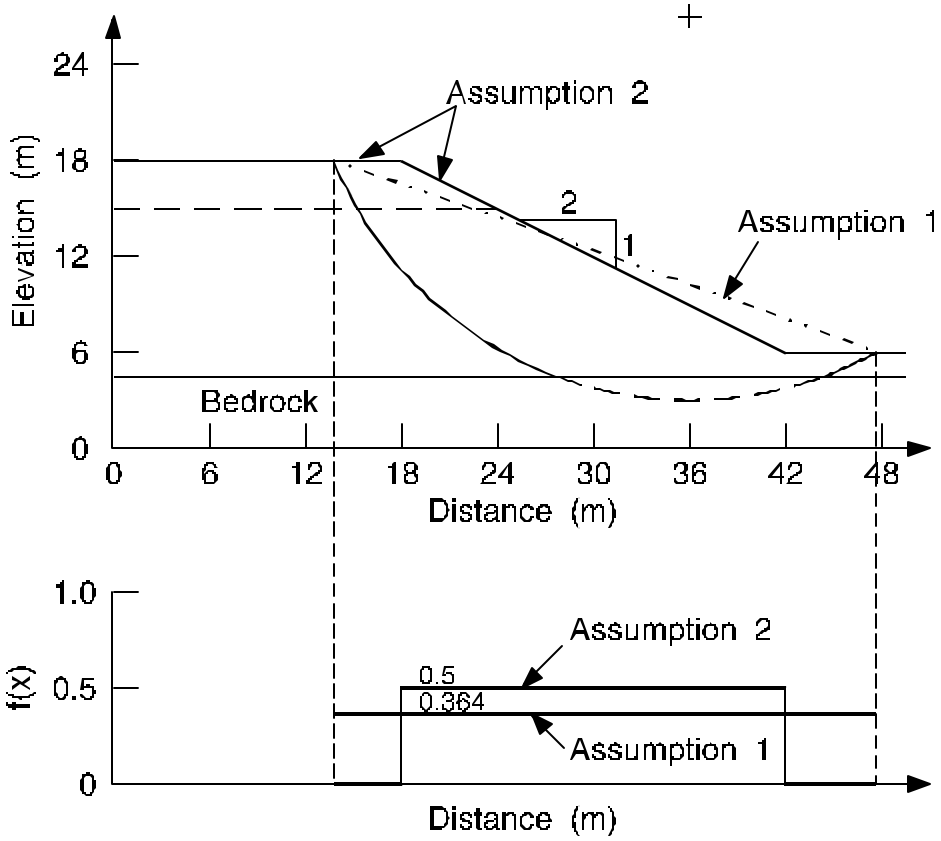
$$E_R = E_L + \frac{(c'b - ub\tan f')\cos\alpha}{F} + N\left(\frac{\tan f'\cos\alpha}{F} - \sin\alpha\right) - kW + [D\cos\omega] \tag{8.16}$$

The interslice normal forces are solved using an integration procedure commencing at the left end of each slip surface.

Corps of Engineers Interslice Force Function

The Corps of Engineers method satisfies only force equilibrium for the overall slope. The direction of the interslice force is assumed to be equal to the average surface slope. This is interpreted as either equal to the average slope between the extreme entrance and exit of the slip surface (Assumption No. 1) or the changing slope of the ground surface (Assumption No. 2, Figure 8.8). In other words, for Assumption No. 2, the slope of the interslice forces changes across the geometry depending on ground surface. In both cases, SLOPE/W generates the interslice force function.

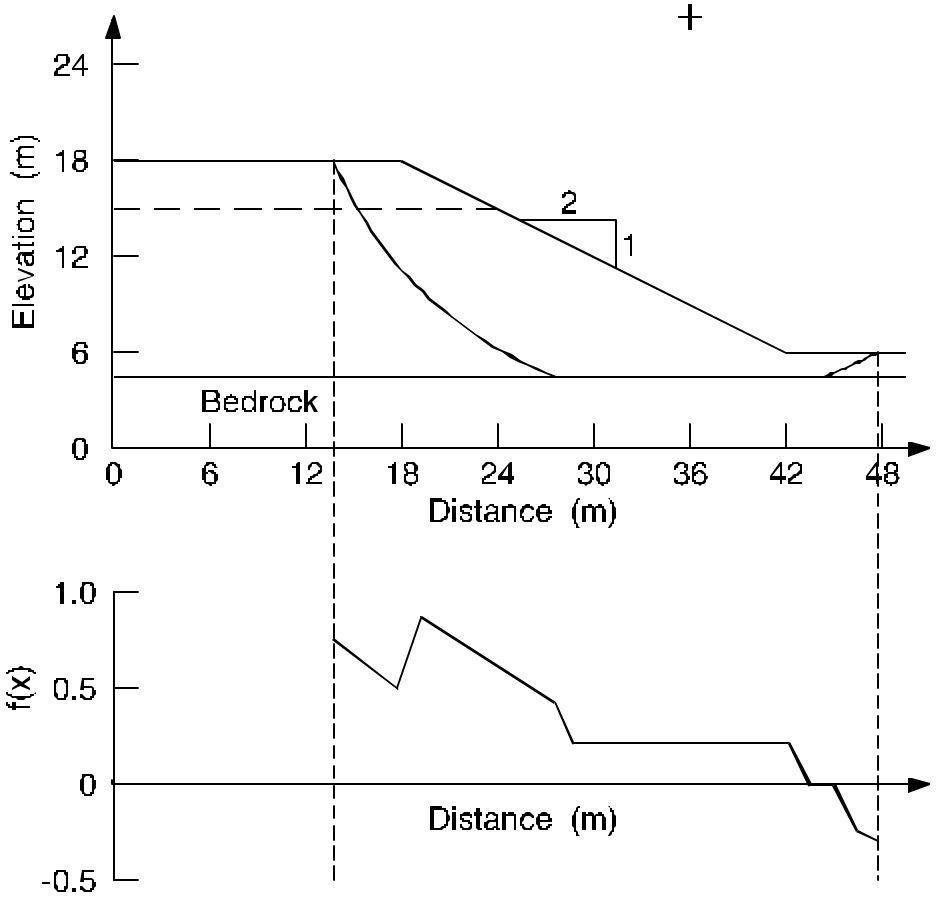
Figure 8.8 Description of the Corps of Engineers Assumptions Regarding Interslice Force Directions



Lowe-Karafiath Interslice Force Function

The Lowe-Karafiath method satisfies only force equilibrium for the overall slope. The direction of the resultant interslice force is assumed to be equal to the average of the ground surface and slip surface slopes. Figure 8.9 shows the function that would be generated for a composite slip surface.

Figure 8.9 The Lowe-Karafiath interslice force direction assumption (composite slip surface)



Fredlund-Wilson-Fan Interslice Force Function

A generalized function has been proposed for the direction of the interslice forces (Fan, Fredlund and Wilson, 1986). The function is based on a two-dimensional finite element analysis of a linear elastic continuum using constant strain triangular elements. The normal stresses in the *x*-direction and the shear stresses in the *y*-direction were integrated along vertical planes within a sliding mass to obtain normal and shear forces, respectively. The ratio of the shear to the normal force was plotted along each vertical section to provide a distribution for the direction of the resultant interslice forces.

The analysis of many slopes showed that the interslice force function could be approximated by an extended form of an error function equation. Inflection points were close to the crest and toe of the slope. The slope of the resultant interslice forces was steepest at the midpoint of the slope and tended to zero at distances back of the crest and beyond the toe. The mathematical form for the empirical interslice form function can be written as follows:

$$f(x) = y e^{-\frac{1}{2}\left(c^n h^n\right)} \tag{8.17}$$

where:

y = the magnitude of the interslice side force ratio at mid slope (i.e., maximum value)
c = a variable that defines the inflection points for each slope angle
n = a power that specifies the flatness or sharpness of curvature of the function
h = the dimensionless position relative to the midpoint of the slope

Figure 8.10 shows the definition of the dimensionless distance, h .

The factor, y , is related to the average inclination of the slope and the depth factor, D_f , for the slip surface under consideration.

$$y = Exp\left[D_i + D_s\left(D_f - 1.0\right)\right] \tag{8.18}$$

where:

D_f = depth factor

D_i = the natural logarithm of the intercept on the ordinate when $D_f = 1.0$

D_s = slope of the depth factor versus ψ relationship for a specific slope

Figure 8.11 defines the depth factor. Further details are presented by Fredlund, Wilson, and Fan.

The parameters y , c, n, and h required to compute the finite element based interslice force function are included in the SLOPE/W software. The only user-defined information required is the location of the crest and toe.

Figure 8.10 Definition of the Dimensionless X-Coordinates for the Finite Element Based Function

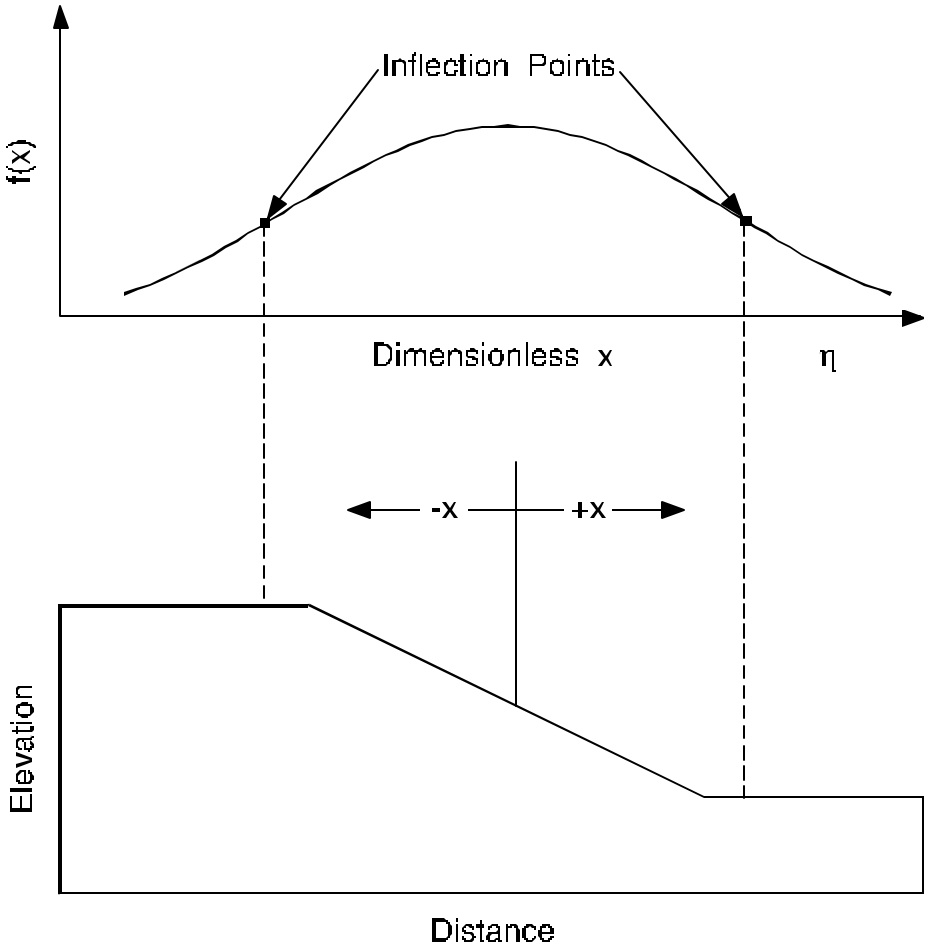
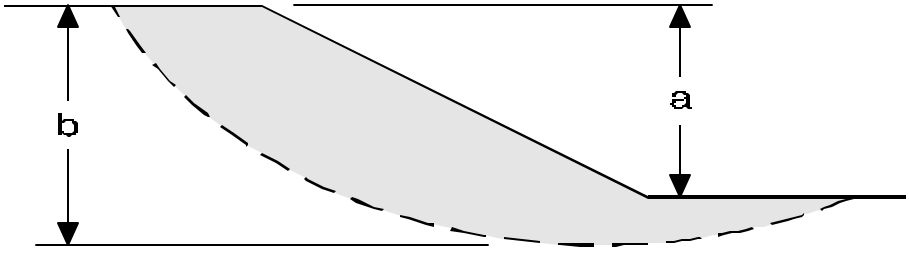


Figure 8.11 Definition of Depth Factor, D_f



$$D_f = \frac{b}{a}$$

Effect Of Negative Pore-Water Pressures

In locations above the groundwater table, the pore-water pressure in a soil is negative relative to the pore-air pressure. This negative pore-water pressure is commonly referred to as the matric suction of the soil. Under negative pore-water pressure conditions the shear strength may not change at the same rate as for total and positive pore-water pressure changes. Therefore, a modified form of the Mohr-Coulomb equation must be used to describe the shear strength of an unsaturated soil (i.e., a soil with negative pore-water pressures):

$$s = c' + (s_n - u_a)\tan f' + (u_a - u_w)\tan f^b \tag{8.19}$$

where:

u_a = pore-air pressure

u_w = pore-water pressure

f^b = an angle defining the increase in shear strength for an increase in matric suction, $(u_a - u_w)$.

Equation 8.19 indicates that the shear strength of a soil can be considered as having three components: the cohesive strength due to *c*', the frictional strength due to f and the suction strength due to f^b.

Factor of Safety for Unsaturated Soil

It is possible to re-derive the above factor of safety equations using the shear strength equation for an unsaturated soil. The mobilized shear force at the base of a slice, S_m, can be written,

$$S_m = \frac{b}{F}\left(c' + (s_n - u_a)\tan f' + (u_a - u_w)\tan f^b\right) \tag{8.20}$$

The normal force at the base of a slice, *N*, is derived by summing forces in the vertical direction:

$$N = \frac{W + (X_R - X_L) - \frac{c'b\sin a}{F} + \frac{u_a\, b\sin a\,(\tan f' - \tan f^b)}{F} + \frac{u_w b \sin a\,\tan f^b}{F}}{\cos a + \frac{\sin a\,\tan f'}{F}} \tag{8.21}$$

For most analyses the pore-air pressure can be set to zero and Equation 8.21 becomes,

$$N = \frac{W + (X_R - X_L) - \frac{c'b\sin a}{F} + \frac{u_w\, b\sin a\,\tan f^b}{F}}{\cos a + \frac{\sin a\,\tan f'}{F}} \tag{8.22}$$

When the soil becomes saturated, f^b can be set to f', and therefore the same equation (i.e., Equation 8.22) can be used for both saturated and unsaturated soils. SLOPE/W uses f^b whenever the pore-water pressure is negative and f' whenever the pore-water pressure is positive.

Two independent factor of safety equations are derived, one with respect to moment equilibrium and the other with respect to horizontal force equilibrium. When only moment equilibrium is satisfied, the factor of safety equation can be written as,

$$F_m = \frac{\sum \left(c' bR + \left[N - u_w b \frac{\tan f^b}{\tan f'} - u_a b \left(1 - \frac{\tan f^b}{\tan f'} \right) \right] R \tan f' \right)}{\sum Wx - \sum Nf} \quad (8.23)$$

The factor of safety equation with respect to horizontal force equilibrium can be written as,

$$F_f = \frac{\sum \left(c' b \cos a + \left[N - u_w b \frac{\tan f^b}{\tan f'} - u_a b \left(1 - \frac{\tan f^b}{\tan f'} \right) \right] \tan f' \cos a \right)}{\sum N \sin a} \quad (8.24)$$

When the pore-air pressure is zero (i.e., atmospheric), the entire pore-air pressure term can be dropped. The above formulations apply for both saturated and unsaturated soils. When the soil is saturated, the f^b term must be set equal to f'.

Use of Unsaturated Shear Strength Parameters

SLOPE/W only considers unsaturated soil shear strength conditions when the pore-water pressures are negative. Under these conditions the angle, f^b, is used to compute the mobilized shear strength force at the base of a slice.

The following types of input data help in understanding how SLOPE/W accommodates unsaturated soil conditions:

1. $f^b = 0.0$. When f^b is left blank or set to 0.0, any negative pore water pressure will be set to zero. There will be no increase in the shear strength due to the negative pore-water pressures (suction). Often the engineer does not want to rely upon any shear strength due to the negative pore-water pressures. In this case, the f^b angles should be set to 0.0.

2. $f^b = f'$. This is an upper limit value for f^b. The input of a value of this magnitude states that negative pore-water pressures will be as effective in increasing the shear strength of a soil as positive pore-water pressures are in reducing the shear strength. This may be reasonable for the saturated capillary zone immediately above the groundwater table. However, the engineer must make the decision whether these negative pore-water pressures are likely to remain near the same magnitudes over the time span of interest.

3. $0.0 < f^b < f'$. This condition assumes that the f^b lies between zero and the effective angle of internal friction. All published research literature has shown this to be the case in laboratory testing programs. Most common values range from 15 to 20 degrees. However, the engineer must again decide whether the negative pore-water pressures are likely to remain near the same magnitudes during the time span under consideration.

When a compacted earth fill is being placed, the pore-air pressure may also increase to values above atmospheric conditions. These can be considered using SLOPE/W by dividing the earthfill into layers and inputting differing pore-air pressures for each layer using the KeyIn Pore Pressure: Air Pressure command.

Solving For The Factors Of Safety

Four different stages are involved in computing the various factors of safety. The following section describes these stages.

Stage 1 Solution

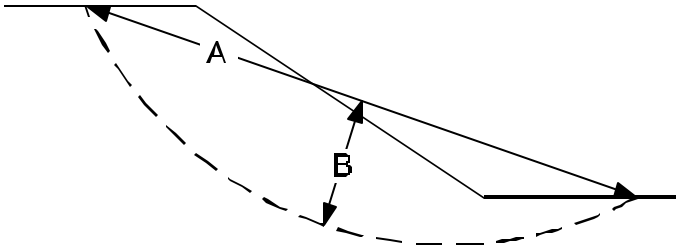
For the first iteration, both the interslice normal and shear forces are set to zero. The resulting moment equilibrium factor of safety is the Ordinary or Fellenius factor of safety. The force equilibrium factor of safety has received little mention in the literature and is of little significance. The first iteration factors of safety are used as approximations for starting the second stage.

Stage 2 Solution

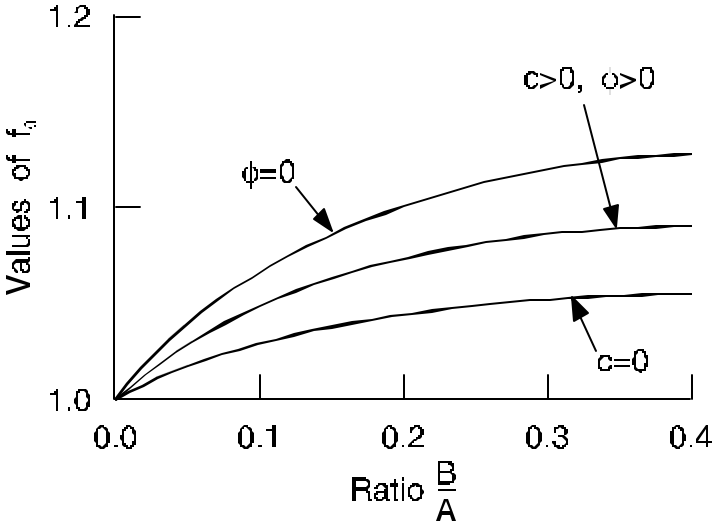
Stage 2 starts the solution of the nonlinear factor of safety equations. Lambda, l , is set to zero and therefore, the interslice shear forces are set to zero. Usually 4 to 6 iterations are required to ensure convergence of the moment and force equilibrium factor of safety equations. The answer from the moment equation corresponds to Bishop's Simplified method. The answer from the force equilibrium equation corresponds to Janbu's Simplified method without the application of the empirical correction factor, f_o. The correction factor is dependent on the shape of the slip surface and the relative amount of cohesion and friction in the soil (Figure 8.12). The computed Janbu factor of safety must be manually adjusted for f_o, if so desired. SLOPE/W does not apply this empirical correction.

Figure 8.12 Correction Factors for Janbu's Simplified Method

(a) 'B' Distance defined by Janbu, Bjerrum and Kjaernsli



(b) Janbu's Simplified Method Correction Factor, f_o



Stage 3 Solution

Except for the GLE method, Stage 3 solution is required for all methods that consider interslice forces. Stage 3 computes the moment and force equilibrium factors of safety for any general interslice force function.

In Stage 3, SLOPE/W computes a lambda, l , that provides an equal value for the force and moment equilibrium factors of safety (i.e., $F_m = 1$). The technique used is called the "Rapid Solver" and is similar in concept to a Newton-Raphson technique.

The Rapid Solver technique works as follows. SLOPE/W computes the initial value for lambda, l , as being equal to 2/3 of the chord slope (Figure 8.13). The moment and force equilibrium factors of safety are computed using this estimate of lambda. These factors of safety along with the factors of safety corresponding to a lambda equal to 0.0 are used to predict a lambda value where the force and moment equilibrium factors of safety will be equal (Figure 8.14).

The above procedure of estimating new lambda values is repeated until the force and moment equilibrium factors of safety are within the selected tolerance.

Any one of the interslice force functions, *f*(*x*), can be used when solving for the factor of safety.

Figure 8.13 Estimate of the Initial Lambda Value for the Rapid Solver

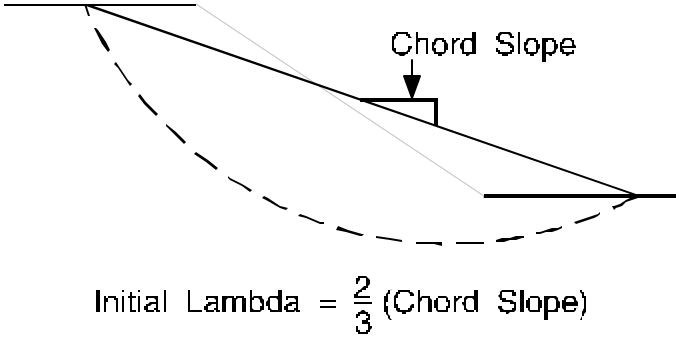
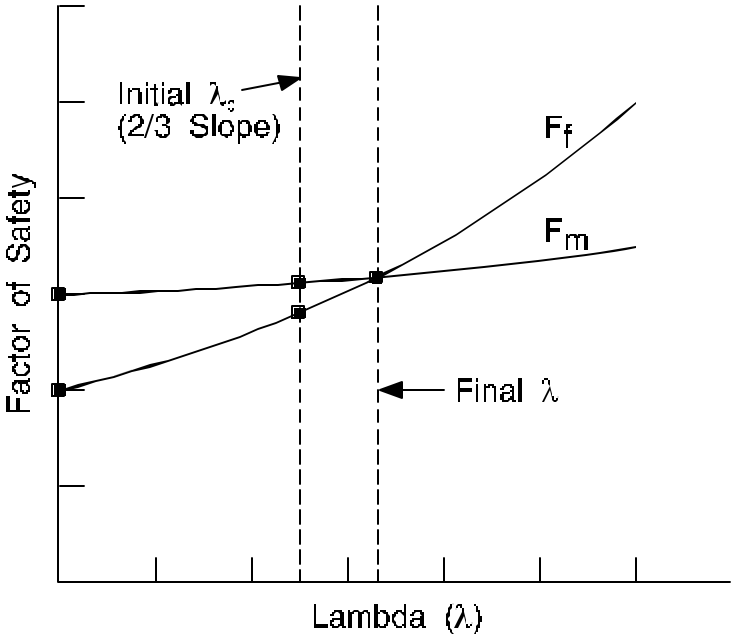
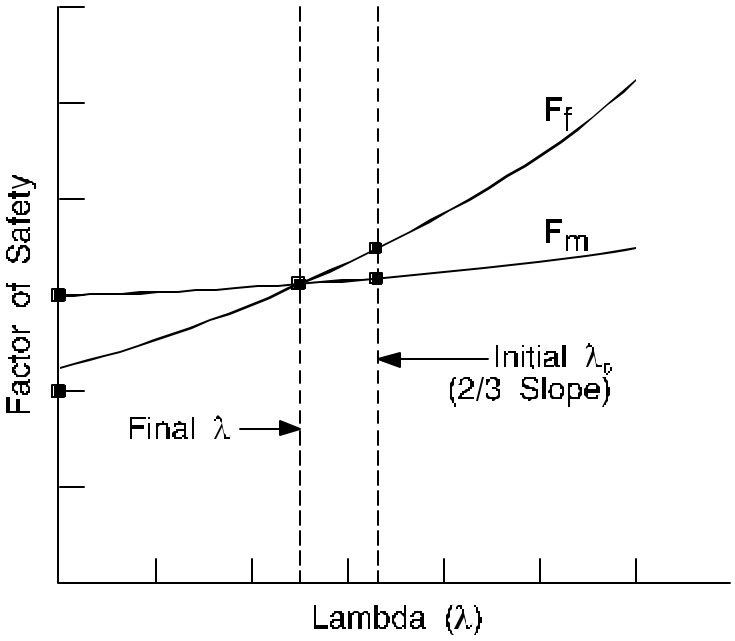


Figure 8.14 Procedure Used by the General Limit Equilibrium Rapid Solver

(a) $F_m > F_f$ **at estimated** l_0



(b) $F_m < F_f$ **at estimated** l_0



Stage 4 Solution

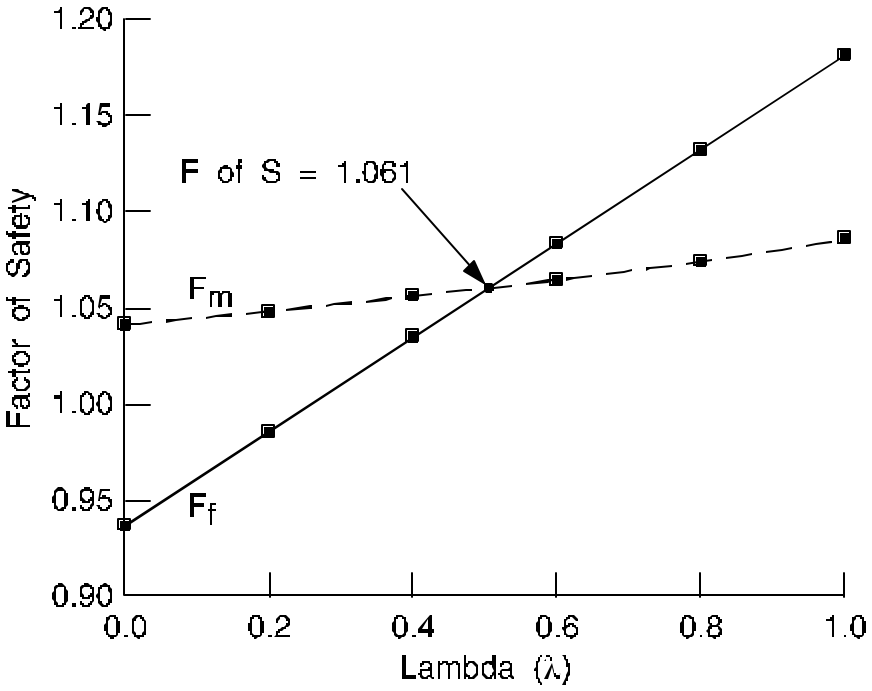
Stage 4 is used when a series of lambda values are selected and the moment and/or force equilibrium factors of safety are solved. Stage 4 is always used for the GLE method of analysis. The factors of safety for various lambda values can be plotted as shown in Figure 8.15. The factor of safety satisfying both moment and force equilibrium can be selected from the plot.

Stage 4 provides a complete understanding of the relationship between the moment and force equilibrium factors of safety for a specific interslice force function. It can be used to simulate essentially all slope stability methods that consider the interslice force function.

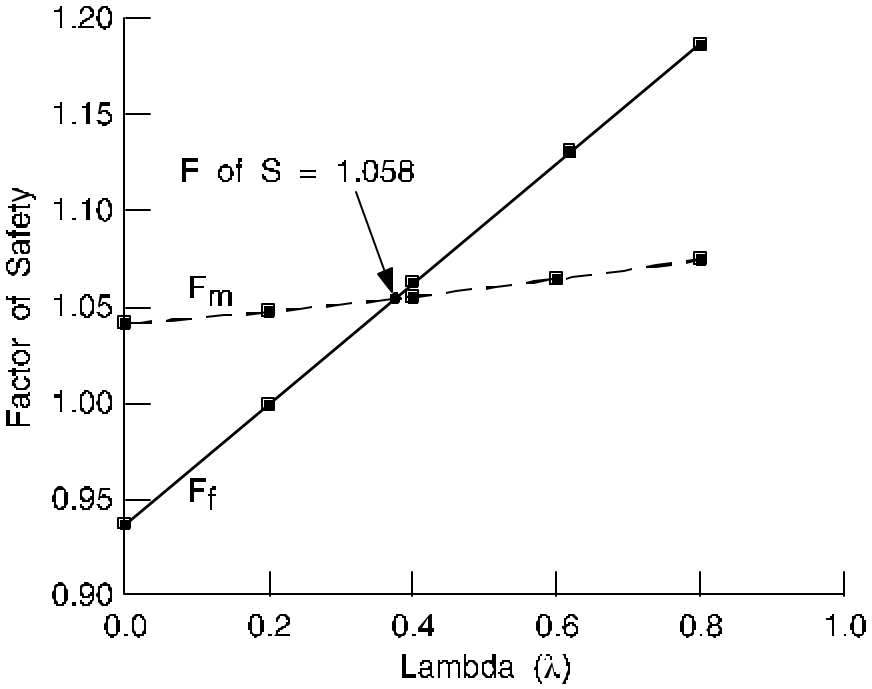
The Stage 4 solution is also used to simulate the Corps of Engineers and Lowe-Karafiath methods of analysis. The factor of safety is computed using the force equilibrium equation for a lambda value of 1.0.

Figure 8.15 Variation of Moment and Force Equilibrium Factors of Safety with respect to Lambda

(a) f(x) = Half-Sine



(b) f(x) = Constant



Simulation of the Various Methods

The General Limit Equilibrium (GLE) formulation and solution can be used to simulate most of the commonly used methods of slices. From a theoretical standpoint, the various methods of slices can be categorized in terms of the conditions of static equilibrium satisfied and the assumption regarding the interslice forces. Table 8.3 summarizes the conditions of static equilibrium satisfied by many of the commonly used methods of slices. Table 8.4 summarizes the assumption used in each of the methods of slices to render the analysis determinate.

Table 8.3 Conditions of Static Equilibrium Satisfied by Various Limit Equilibrium Methods

	Force Equilibrium		
Method	**1st Direction* (e.g., Vertical)**	**2nd Direction* (e.g., Horizontal)**	**Moment Equilibrium**
Ordinary or Fellenius	Yes	No	Yes
Bishop's Simplified	Yes	No	Yes
Janbu's Simplified	Yes	Yes	No
Janbu's Generalized	Yes	Yes	**
Spencer	Yes	Yes	Yes
Morgenstern-Price	Yes	Yes	Yes
GLE	Yes	Yes	Yes
Corps of Engineers	Yes	Yes	No
Lowe-Karafiath	Yes	Yes	No

* Any of two orthogonal directions can be selected for the summation of forces.

** Moment equilibrium is used to calculate interslice shear forces.

Table 8.4 Assumptions Used in Various Limit Equilibrium Methods

Method	Assumption
Ordinary or Fellenius	Interslice forces are neglected.
Bishop's Simplified	Resultant interslice forces are horizontal (i.e., there are no interslice shear forces).
Janbu's Simplified	Resultant interslice forces are horizontal. An empirical correction factor, f_o, is used to account for interslice shear forces.
Janbu's Generalized	Location of the interslice normal force is defined by an assumed line of thrust.
Spencer	Resultant interslice forces are of constant slope throughout the sliding mass.
Morgenstern-Price	Direction of the resultant interslice forces is determined using an arbitrary function. The percentage of the function, l, required to satisfy moment and force equilibrium is computed with a rapid solver.
GLE	Direction of the resultant interslice forces is defined using an arbitrary function. The percentage of the function, l, required to satisfy moment and force equilibrium is computed by finding the intersecting point on a factor of safety versus Lambda plot.
Corps of Engineers	Direction of the resultant interslice force is: i) equal to the average slope from the beginning to the end of the slip surface or ii) parallel to the ground surface.
Lowe-Karafiath	Direction of the resultant interslice force is equal to the average of the ground surface and the slope at the base of each slice.

Table 8.5 shows the procedure for simulating various methods of slices when using SLOPE/W. Figure 8.16 graphically shows the relationship between the various methods of slices.

Table 8.5 Simulation of Commonly Used Methods of Slices

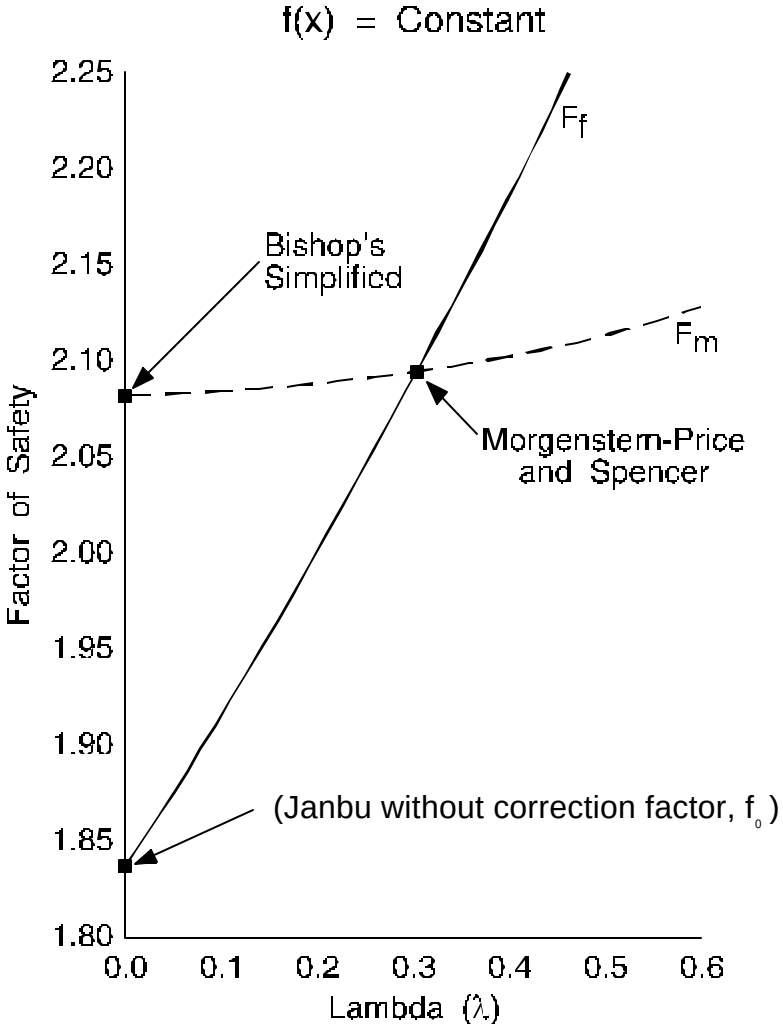
Method of Slices	Stage	Interslice Force Function	Lambda
Fellenius or Ordinary	1	N.A.	N.A. (Set to 0.0)
Bishop's Simplified	2	N.A.	0.0
Janbu's Simplified*	2	N.A.	0.0
Janbu's Generalized**	N.A.	N.A.	N.A.
Spencer	3	$f(x) = 1.0$	Computed
Morgenstern-Price	3	Any $f(x)$	Computed
GLE	4	Any $f(x)$	User-defined and Computed
Corps of Engineers	4	As illustrated on Figure 8.8	$l = 1.0$
Lowe-Karafiath	4	As illustrated on Figure 8.9	$l = 1.0$

N.A. Not Applicable.

* Must be multiplied by correction factor, f_o.

** Cannot be simulated

Figure 8.16 Comparison of Factors of Safety Obtained by Various Methods of Analysis



Spline Interpolation of Pore-Water Pressures

A spline interpolation technique is used to determine the pore-water pressure at the base of a slice when the pore-water pressures are defined at discrete points.

The technique involves the fitting of a spline function to a series of spatially distributed points. The fitting of the function to the points results in the calculation of weighting coefficients. The weighting coefficients can then be used to compute values for any other point in the region. Although the solving of a large problem using this technique requires considerable computer storage, it has been found that a small number of designated points can provide reasonably accurate results.

To illustrate the spline interpolation technique, consider the following two-dimensional problem. Suppose we know a set of values, u_i, at *N* given points $(x_i, y_i \text{ with } i = 1, N)$, and we want to estimate the value of *u* at some other points, *M*(*x*,*y*).

Let:

$$u(x,y) = P(x,y) + \sum_{i=1}^{N} l_i K\left(h_m - h_i\right) \tag{8.25}$$

where:

P(x,y) = the chosen trend,
K(h) = the chosen interpolation function,
h = the distance between two points, $(\text{e.g., } h = h_m - h_i)$, where:

$$(h_m - h_i) = (x_m - x_i)^2 + (y_m - y_i)^2$$

$l\,i$ = the computed weighting coefficients referred to as Kriging coefficients.

In the SLOPE/W formulation,

$$P(x,y) = a + bx + cy \tag{8.26}$$

-- and --

$$K(h) = d(0) + h^2 \log h \tag{8.27}$$

where $d(0)$ is the nugget effect. This will be explained later and for the present is assumed to be zero.

The weighting coefficients $(a, b, c, l_1, l_2, \ldots. l_N)$ are the solution of the following set of linear equations:

$$\begin{bmatrix} K(0) \ldots & K(h_1 - h_n) & 1 & x_1 & y_1 \\ & . & . & . & . \\ & & 1 & x_N & y_N \\ & & 0 & 0 & 0 \\ & & & 0 & 0 \\ & & & & 0 \end{bmatrix} \begin{bmatrix} l_1 \\ . \\ . \\ a \\ b \\ c \end{bmatrix} = \begin{bmatrix} u_1 \\ . \\ u_N \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

The above system of linear equations is solved for the weighting coefficients. The value of $u(x,y)$ can now be computed at any point, x,y using the equation:

$$u(x,y) = a + bx + cy + \sum_{i-1}^{N} \left(l_i (h_m - h_i)^2 \log(h_m - h_i) \right) \tag{8.28}$$

The following properties can be derived from Equation 8.28:

- At a point x_1, y_1, if a(0) = 0 and $K(0) = 0$), then,

  $$u(x_1, y_1) = u_1$$

- If for the point x_1, y_1, $K(0) = a_1$ and $u(x_1, y_1) \neq u_1$, then,

  $$d(0) = d(1) \neq 0$$

Therefore, by selecting different nugget values for the initial points, it is possible to help the estimated values coincide with the initial values. At its limit, if $d(0)$ is the same for all points and its value becomes large,

$$u(x,y) = P(x,y) = a + bx + cy \tag{8.29}$$

This is equivalent to the least square solution of fitting.

Let us select the function, $K(h) = h^2 \log h$ in Equation 8.27. The solution of this spline problem can be visualized as a thin plate deforming in such a way as to pass through the deflection, u_i, at all points, x_1, y_1.

Finite Element Pore-Water Pressure

SLOPE/W can use the pore-water pressure computed by a SEEP/W or SIGMA/W finite element analysis.

To determine the pore-water pressure at the base of a slice from finite element results, SLOPE/W uses the same interpolation shape functions as used in the finite element formulation.

In the SEEP/W formulation, the hydraulic head distribution within an element in terms of the heads at the element nodes is expressed by the equation,

$$h = <N> \{H\} \tag{8.30}$$

where:

h = head anywhere in the element
$<N>$ = matrix of interpolation functions
$\{H\}$ = matrix of heads at the element nodes

When using this technique, SLOPE/W finds the element that exists at the center of the slice base. SLOPE/W then finds the corresponding local coordinates of the center of the slice base, and the matrix of the interpolation functions. Finally, SLOPE/W computes the pore-water pressure at this location from the nodal pore-water pressure conditions of the element nodes.

Slice Width

SLOPE/W analyzes each potential slip surface by dividing it into sections and then dividing each section into one or more slices. The width of each slice is variable.

Figure 8.17 and 8.18 show how the potential sliding mass is divided into sections. First, SLOPE/W computes the left and right intercepts. Then, moving from left to right, SLOPE/W breaks the slip surface into sections, based on changes in geometry or stratigraphy and based on where the slip surface intersects the soil boundaries. For fully specified or block specified slip surfaces, sections are defined at each break in the slip surface. For cases with pressure boundaries, the breaking points of the pressure boundaries are considered as separate sections.

Once the left and right intercepts are computed, SLOPE/W computes an average slice width based on the number of slices defined for the problem. In equation form,

$$W_a = \frac{x_R - x_L}{n} \tag{8.31}$$

where:

W_a = average slice width
x_R = x-coordinate of right intercept
x_L = x-coordinate of left intercept
n = number of slices (user-specified)

The number of slices in each section is taken as the integer value that results from dividing the section width by the average slice width. The slice width in the section is then computed as the section width divided by the number of slices in the section.

In certain cases, SLOPE/W may add several slices to the user-specified number of slices in order to maintain the system of subdividing the potential sliding mass. For example, each section must have a minimum of one slice. In some cases, if a section is narrower than 1% of the average width of the slice, the section will be added to the adjacent section. This is to prevent the existence of a slice with extremely small width.

The system of dividing the sliding mass into sections and then slices results in the factor of safety being insensitive to the slice width.

Figure 8.17 Definition of Sections for a Composite Slip Surface

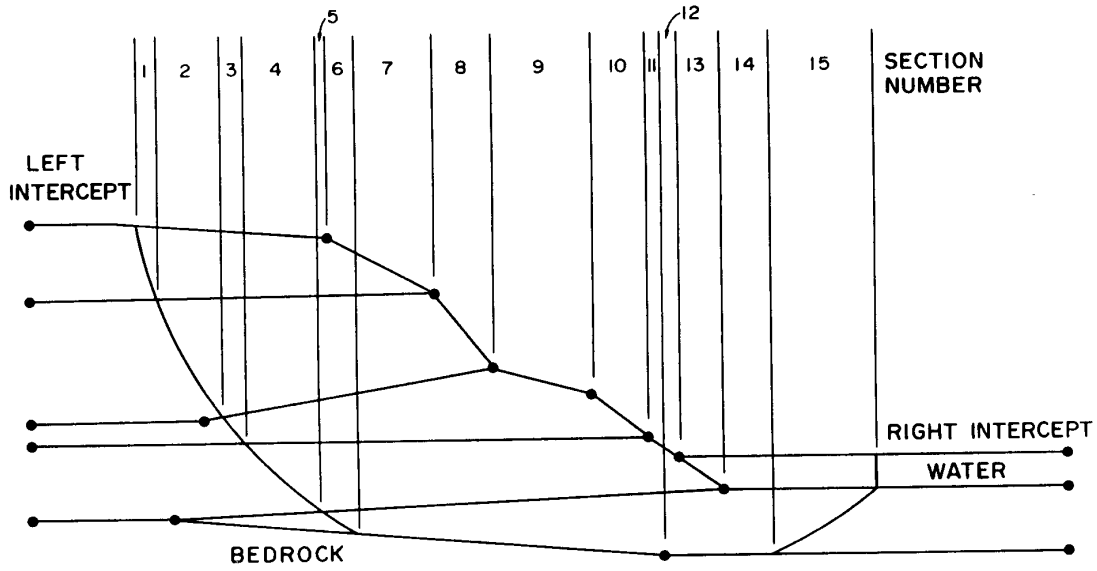
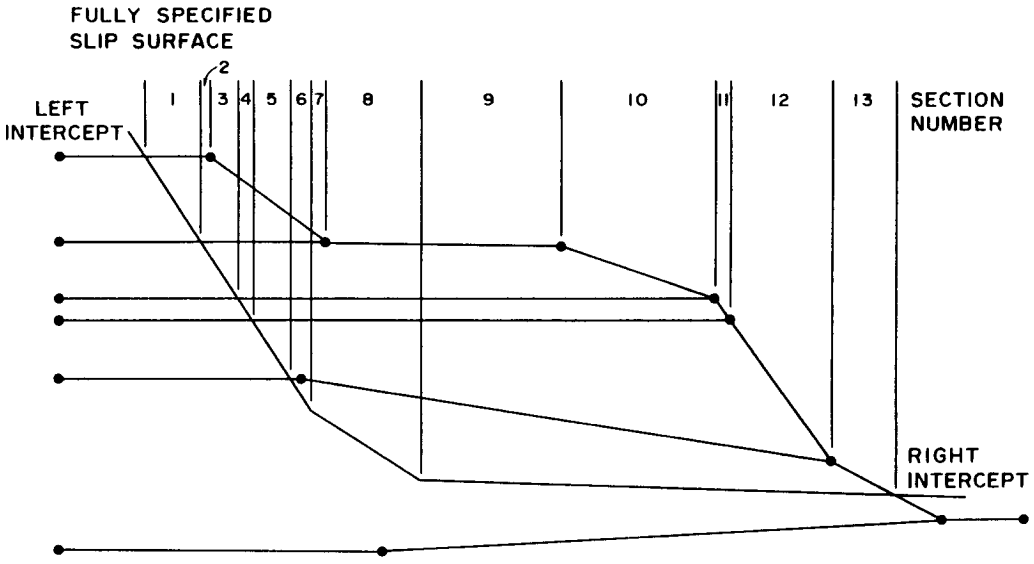


Figure 8.18 Definition of Sections for a Fully Specified Slip Surface



Moment Axis

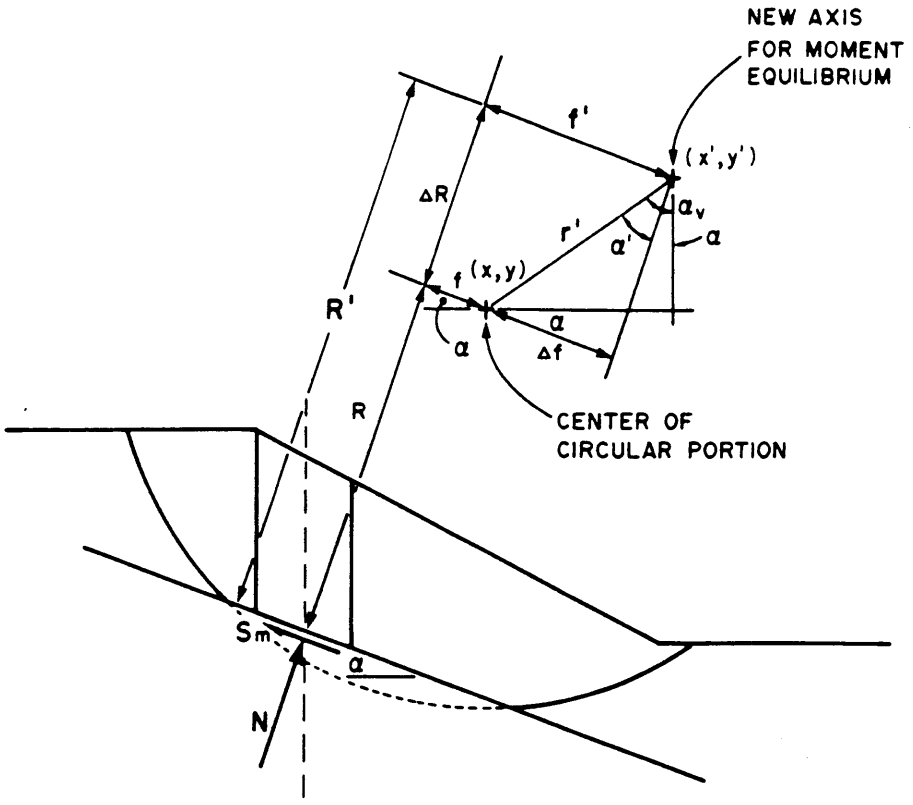
When the grid and radius method is selected using KeyIn Analysis Control to define the slip surfaces, the moment factor of safety is computed by summing moments about each grid point. However, it is possible to use one single point at which to sum moments for all slip surfaces. This point is known as the axis. The grid point is used to define the shape of the slip surface, and the axis point is used for summing moments.

Figure 8.19 shows the adjustments that are made to the radius when the grid rotation point and axis point are different.

The position of the moment center has a negligible effect on factors of safety computed by methods that satisfy both force and moment equilibrium (e.g., the GLE, the Morgenstern-Price and the Spencer methods). The factor of safety can be slightly affected by the position of the moment axis when the slip surface is non-circular and the method satisfies only force or only moment equilibrium (Fredlund, Zhang and Lam, 1992).

As a general rule, the axis point should be located approximately at the center of rotation of the slip surfaces.

Figure 8.19 Definition of Variables Associated with an Independent Moment Equilibrium Axis



$$r' = \sqrt{(x' - x)^2 + (y' - y)^2}$$

$$a_v = \arctan\left(\frac{x' - x}{y' - y}\right)$$

$$a' = a_v - a$$

$$\Delta R = r' \cos a'$$

$$\Delta f = r' \sin a'$$

$$f = f + \Delta f$$

$$R' = r + \Delta R$$

Soil Strength Models

Anisotropic Strength

Anisotropic strength can be simulated with the KeyIn Soil Properties command by specifying the shear strength parameters (c and ϕ) separately for the horizontal and vertical directions. The c and ϕ values at the base of each slice used in the strength calculation is adjusted based on the inclination angle as:

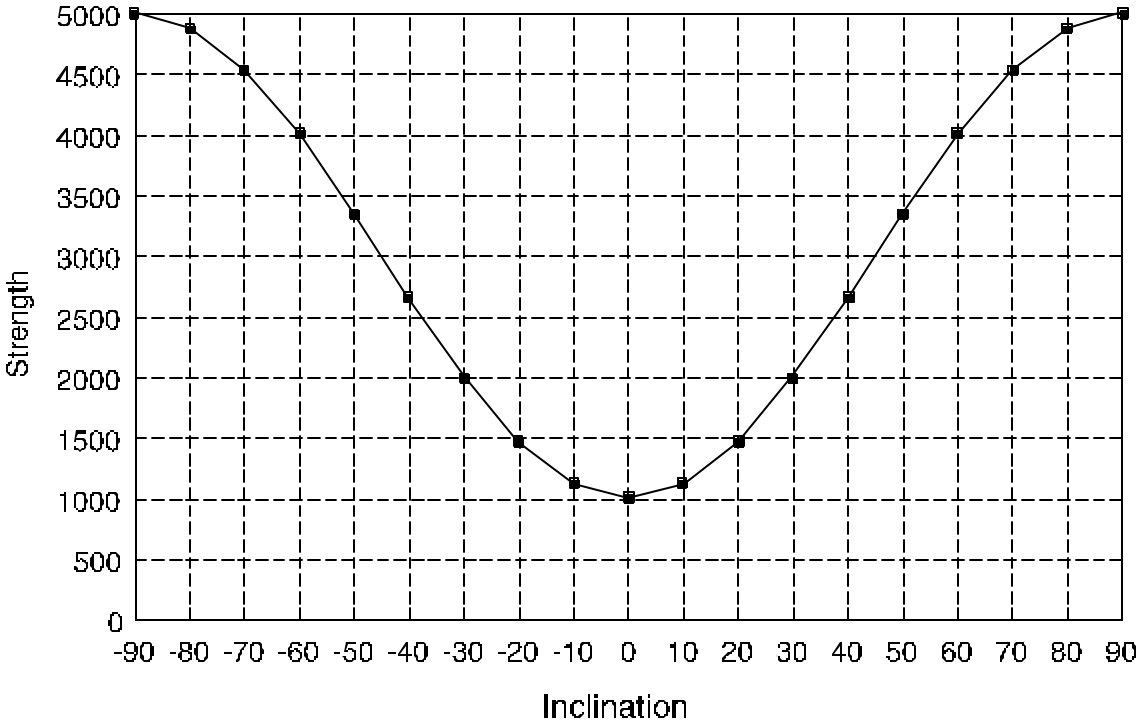
$$\begin{aligned} c &= c_x \cos^2 \alpha + c_y \sin^2 \alpha \\ \phi &= \phi_x \cos^2 \alpha + \phi_y \sin^2 \alpha \end{aligned} \qquad (8.32)$$

where:

c = adjusted cohesion along the slice base
c_x = specified horizontal cohesion
c_y = specified vertical cohesion
α = slice base inclination angle
ϕ = adjusted frictional angle along the slice base
ϕ_x = specified horizontal frictional angle
ϕ_y = specified vertical frictional angle

Figure 8.20 shows an example of the variation in strength with the base inclination angle when the shear strength is 1000 and 5000 in the horizontal and vertical directions respectively.

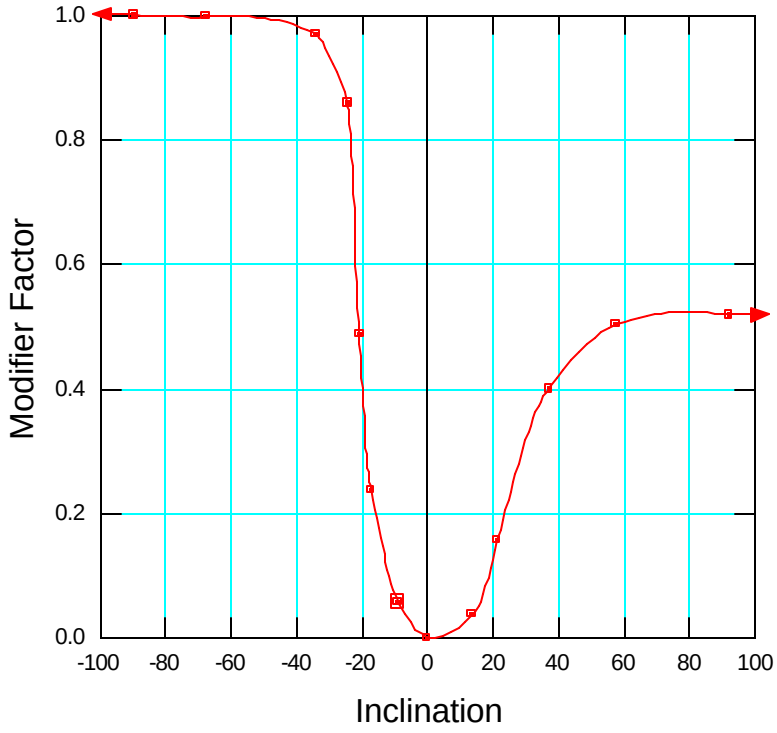
Figure 8.20 Variation in Anisotropic Strength Parameters with the Base Inclination Angle



Anisotropic Strength Modifier Function

Anisotropic strength can be simulated by modifying the shear strength at the base of each slice with a modifier function. The modifier function can be defined as a general data-point function using the KeyIn Strength Functions: Anisotropic command, as illustrated in Figure 8.21. For example, if the base angle is -20 degrees, the modifier factor will be 0.4 and shear strength at the base of the slice will be 40% of the maximum strength.

Figure 8.21 Typical Example of an Anisotropic Strength Function

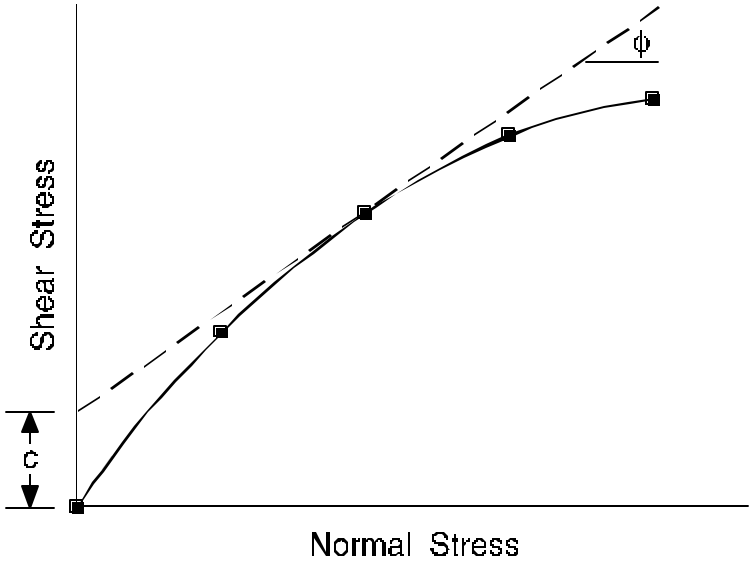


Positive angles are for base inclinations where the potential slice movement is down-slope, and negative angles are for base inclinations where the potential slice movement is up-slope.

Shear/Normal Strength Function

The failure envelope can be defined as a general data point function, as illustrated in Figure 8.22, by choosing KeyIn Strength Functions: Shear/Normal.

Figure 8.22 Definition of Cohesion and Phi by a Shear/Normal Strength Function



SLOPE/W computes a slope (f) and *y*-intercept (*c*) for each slice as a function of the normal stress at the slice base. Consequently, c and f may vary for each slice. At normal stresses greater than the last data point, SLOPE/W uses the *c* and f values at the last data point. The shear/normal strength function must always start at the origin (0,0).

Finite Element Stress Method

In addition to the limit equilibrium methods of analysis, SLOPE/W also provides an alternative method of analysis using the stress state obtained from SIGMA/W, a GEO-SLOPE program for stress and deformation analysis. The following sections outline the theoretical basis and the solution procedures used by the SLOPE/W Finite Element Stress method.

Stability Factor

As mentioned earlier in the Definition of Variables section, the factor of safety obtained using a limit equilibrium method is defined as that factor by which the shear strength of the soil must be reduced in order to bring the mass of soil into a state of limiting equilibrium along a selected slip surface. Furthermore, due to the nature of the method, the following two assumptions are made with respect to the factor of safety:

- The factor of safety of the cohesive component of strength and the frictional component of strength are equal for all soils involved.
- The factor of safety is the same for all slices.

The above assumptions are no longer necessary in the finite element stress method. In other words, the computed "factor of safety" using the finite element stress approach is not the same factor of safety as in the limit equilibrium approach. To preserve the original meaning of the factor of safety, the "factor of safety" computed using the Finite Element Stress method is referred to as the *stability factor* in SLOPE/W.

The stability factor (S.F.) of a slope by the finite element stress method is defined as the ratio of the summation of the available resisting shear force along a slip surface $\left(\sum S_r\right)$ to the summation of the mobilized shear force along a slip surface $\left(\sum S_m\right)$. In equation form, the stability factor (S.F.) is expressed as:

$$S.F. = \frac{\Sigma S_r}{\Sigma S_m} \tag{8.33}$$

The available resisting force of each slice is calculated by multiplying the shear strength of the soil at the base center of the slice with the base length. Therefore, from the modified form of the Mohr-Coulomb equation for an unsaturated soil, as presented in the Effect of Negative Pore-Water Pressures section, the available resisting force is:

$$S_r = sb = \left(c' + \left(s_n - u_a\right)\tan f' + \left(u_a - u_w\right)\tan s^b\right)b \tag{8.34}$$

where:

s = effective shear strength of the soil at the base center of a slice
b = base length of a slice
s_n = normal stress at base center of a slice

Similarly, the mobilized shear force of each slice is calculated by multiplying the mobilized shear stress $\left(t_m\right)$ at the base center of the slice with the base length.

$$S_m = t_m b \tag{8.35}$$

A local stability factor of a slice can also be obtained when the available resisting shear force of a slice is compared to the mobilized shear force of a slice.

$$Local\ S.F. = \frac{S_r}{S_m} = \frac{sb}{tb} \tag{8.36}$$

Of significance is the fact that both the normal stress (s_n) and the mobilized shear stress (t_m) are computable values from a SIGMA/W analysis. Therefore, the equations for computing the stability factors are linear; that is, no iteration is required to establish the stability factors as in the limit equilibrium method. Iterations may be required in the SIGMA/W analysis but not in the SLOPE/W analysis.

Normal Stress and Mobilized Shear Stress

To use the Finite Element Stress method, you need to start by performing a SIGMA/W analysis. The procedures for doing this are provided in the SIGMA/W User's Guide.

The information required from the SIGMA/W analysis is the stress state as describe by $s_x, s_{y,}$ and t_{xy} at each Gauss point within each element. These stress values are used to compute the normal stress and the mobilized shear stress at the base center of each slice. The procedure is as follows:

Step 1: Identification of Element

The first step is to identify the element that contains the base center of a slice. This is done by first finding the global coordinates of the base center, then solving for the local coordinates (*r*, *s*) of the base center with respect to the elements.

In a finite element analysis, the global coordinates of the base center can be related to the global coordinates of the nodal points of any element by:

$$x = <N> \{X\} \tag{8.37}$$

$$y = <N> \{Y\} \tag{8.38}$$

where:

x = the global x coordinate of the base center
y = the global y coordinates of the base center
$\{X\}$ = global x coordinates of the element nodal points
$\{Y\}$ = global y coordinates of the element nodal points
$<N>$ = matrix of interpolation functions

Since the interpolation functions $<N>$ are defined in terms of the local coordinates (r,s) and the global coordinates are known, the local coordinates of the base center within an element can be obtained by solving the above two simultaneous equations.

A base center is within an element if the local coordinates of the base center lie within the following ranges:

- For a triangular element: $(0 \le r \ge 1)$ and $(0 \le s \ge 1)$
- For a rectangular element: $(-1 \le r \ge 1)$ and $(-1 \le s \ge 1)$

If the local coordinates are outside these ranges, the base center is not within the element, and the procedures moves on to the next element. This continues until the element that encompasses the base center is found.

Step 2: Element Nodal Stresses

SIGMA/W calculates and stores the computed stresses at element Gauss points. To compute the stress state at the slice base center, it is first necessary to establish the stress state at the element nodes. This is done by projecting the Gauss values to the nodes and then averaging the nodal values obtained from each adjoining element.

The projection is done with the use of the interpolating functions. In equation form,

$$f = \langle N \rangle \{F\} \tag{8.39}$$

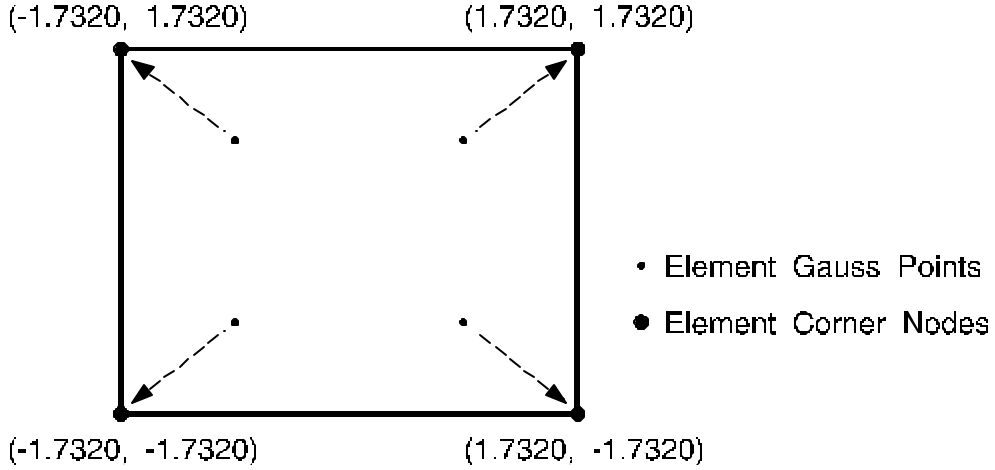
where:

f = stress at the element nodes
$\langle N \rangle$ = matrix of the interpolating functions
$\{F\}$ = stress values at the Gauss points

The interpolating functions are the same as the standard functions used to describe a variable within an element in terms of nodal values, except that the local coordinates are the reciprocal of the standard Gauss point integration points.

Consider, for example, the local coordinates of a Gauss integration point inside an element are (0.577, 0.577). When projecting outwards from the Gauss point to the corner node, the local coordinates for the closest corner node are (1.73, 1.73). Figure 8.23 illustrates this projection scheme for a quadrilateral element.

Figure 8.23 Local Coordinates at the Corner Nodes of an Element with Four Integration Points



The above projection is carried out for each element in a problem, and the values from each adjoining element are then averaged. Upon completion of this procedure, $\sigma_x, \sigma_y,$ and τ_{xy} are known at each node in the entire mesh.

Step 3: Base Center Stresses

Once $\sigma_x, \sigma_y,$ and τ_{xy} are known at the nodes, the same standard set of interpolation functions are used again to calculate stress at the center of the slice base. The local coordinates at the base center are known from element identification calculations (Step 1). In equation form,

$$\{\sigma\} = \langle N \rangle \{\sigma_n\} \tag{8.40}$$

where:

$\{s\}$ = stresses at the base center

<*N*> = matrix of the interpolating functions

$\{s_n\}$ = stresses at the element nodes

At the completion of this step, $s_x, s_{y,}$ and t_{xy} are known at the base center of a slice.

Step 4: Base Normal and Shear Stresses

The normal stress $\{s_n\}$ and the mobilized shear stress (t_m) at the base center are computed using the following equations (Higdon, 1978):

$$s_n = \frac{s_x + s_y}{2} + \frac{s_x - s_y}{2}\cos 2q + t_{xy}\sin 2q \tag{8.41}$$

$$t_m = t_{xy}\cos 2q - \frac{s_x - s_y}{2}\sin 2q \tag{8.42}$$

where:

s_x = total stress in *x*-direction at the base center

s_y = total stress in *y*-direction at the base center

t_{xy} = shear stress in *x*- and *y*-directions at the base center

q = angle measured from positive *x*-axis to the line of application of the normal stress

The line of application of the normal stress is perpendicular to the base plane of the slice, whereas the line of application of the mobilized shear stress is parallel to the base plane.

Important factors to be considered when applying the Finite Element Stress method are discussed in Finite Element Stress Method in Chapter 7.

Probabilistic Slope Stability Analysis

Deterministic slope stability analyses compute the factor of safety based on a fixed set of conditions and material parameters. If the factor of safety is greater than unity, the slope is considered to be stable. On the contrary, if the factor of safety is less than unity, the slope is considered to be unstable or susceptible to failure. Deterministic analyses suffer from limitations such as the variability of the input parameters is not considered and questions like "How stable is the slope?" cannot be answered.

Probabilistic slope stability analysis allows for the consideration of variability in the input parameters and it quantifies the probability of failure of a slope. SLOPE/W Version 4 can perform probabilistic slope stability analyses using the Monte Carlo method.

Monte Carlo Method

The Monte Carlo method is a simple but versatile computational procedure that is extremely suitable for a high speed computer. In general, the implementation of the method involves the following (Yang, Fredlund and Stolte, 1993):

- the selection of a deterministic solution procedure, such as the Spencer's method or the finite element stress method.
- decisions regarding which input parameters are to be modelled probabilistically and the representation of their variability in terms of a normal distribution model using the mean value and standard deviation.
- the estimation of new input parameters and the determination of new factors of safety many times.
- the determination of some statistics of the computed factor of safety, the probability density and the probability distribution of the problem.

In SLOPE/W, the critical slip surface is first determined based on the mean value of the input parameters using any of the limit equilibrium and finite element stress methods. Probabilistic analysis is then performed on the critical slip surface, taking into consideration the variability of the input parameters. The variability of the input parameters is assumed to be normally distributed with user-specified mean values and standard deviations.

During each Monte Carlo trial, the input parameters are updated based on a normalized random number. The factors of safety are then computed based on these updated input parameters. By assuming that the factors of safety are also normally distributed, SLOPE/W determines the mean and the standard deviations of the factors of safety. The probability distribution function is then obtained from the normal curve.

The number of Monte Carlo trials in an analysis is dependent on the number of variable input parameters and the expected probability of failure. In general, the number of required trials increases as the number of variable input increases or the expected probability of failure becomes smaller. It is not unusual to do thousands of trials in order to achieve an acceptable level of confidence in a Monte Carlo probabilistic slope stability analysis (Mostyn and Li, 1993).

Parameter Variability

Soils are naturally formed materials, consequently their physical properties vary from point to point. This variation occurs even in an apparently homogeneous layer. The variability in the value of soil properties is a major contributor to the uncertainty in the stability of a slope. Laboratory results on natural soils indicate that most soil properties can be considered as random variables conforming to the normal distribution function (Lumb, 1966, Tan, Donald and Melchers, 1993).

In SLOPE/W, the variability of the input parameters is assumed to be normally distributed. The variability of the following input parameters can be considered:

- material parameters for the various material strength models, including unit weight, cohesion and frictional angles,
- pore-water pressure conditions,
- the magnitude of the applied line loads, and
- the horizontal and vertical seismic coefficients.

Normal Distribution Function

A normal distribution function, often referred to as the Gaussian distribution function, is the most commonly used function to describe the variability of input parameters in probabilistic analyses. The normal distribution is so prevalent because many physical measurements provide frequency distributions that closely approximate a normal curve. A normal distribution function can be represented mathematically as:

$$f(x) = \frac{1}{s\sqrt{2p}} e^{-\frac{(x-m)^2}{2s^2}} \quad -\infty < x < \infty \tag{8.43}$$

where:

$f(x)$ = relative frequency
s = standard derivation
m = mean value

A normal curve is bell shaped, symmetric and with the mean value exactly at middle of the curve. A normal curve is fully defined when the mean value, m and the standard deviation, s are known.

Theoretically, the normal curve will never touch the *x*-axis, since the relative frequency, *f*(*x*) will be nonzero over the entire range from $-\infty$ to $+\infty$. However, for practical purposes, the relative frequency can be neglected after ± 5 standard deviation, s , away from the mean value.

The entire area under the normal curve is equal to unity, and therefore, the area under the curve for a particular range of *x* values represents the probability of obtaining the value within that range. A normal curve has a convenient property that the area under the normal curve between the mean and any point depends only on the number of standard deviations away from the mean. For example, the area or probability of a value, *x*, lying between $\pm 1s$ is 68.26%, between $\pm 2s$ is 95.44%, between $\pm 3s$ is 99.72%, between $\pm 4s$ is 99.99% and between $\pm 5s$ is approximately 100.00%.

For example, in SLOPE/W, the mean cohesion of a soil may be specified as 30 kPa with a standard derivation of 5 kPa. This means that for 100 samples, 68.26% of the samples will have a value between 25 and 35 kPa, and 95.44% of the samples will have a value between 20 and 40 kPa.

Random Number Generation

Fundamental to the Monte Carlo method are the randomly generated input parameters that are fed into a deterministic model. In SLOPE/W, this is done using a random number generation function. To ensure that a new set of random numbers is generated every time SLOPE/W is executed, the random number function is seeded with the current time of the computer clock.

The random numbers generated from the function are uniformly distributed with values between 0 and 1.0. In order to use the uniformly generated random number in the calculations of the normally distributed input parameters, it is necessary to transform the uniform random number to a normally distributed random number. This “normalization” process is done using the following transformation equation as suggested by Box and Muller, (1958):

$$N = \sqrt{(-2\ln R_1)} * \sin(2p\ R_2) \tag{8.44}$$

where :

N = normalized random number
R_1 = uniform random number 1
R_2 = uniform random number 2

The transformation equation requires the generation of two uniform random numbers. The normalized random number can be viewed as the standard normal deviate in a normal curve with a mean value of 0 and standard derivation of 1.

Estimation of Input Parameters

At the beginning of each Monte Carlo trial, all variable input parameters including shear strength parameters, line load magnitudes, seismic coefficients and pore-water conditions are re-evaluated based on the specified mean value, m, the standard deviation, s, and the normalized random number, N. In SLOPE/W, the equation for updating the parameters is:

$$P = m + Ns \tag{8.45}$$

where P is the new trial value of any of the parameters specified with a standard deviation.

For example, consider a line load with a specified mean value of 300 kN and a standard deviation of 25 kN. In a particular Monte Carlo trial, if the normalized random number is -2.0, the trial line load magnitude will be 250 kN.

Equation 8.45 is used to estimate the new shear strength input parameters, new line load magnitudes, new seismic coefficients and new pore-water head. However, in the case of pore-water heads, the new pore-water heads are restricted not to be higher than the ground surface of the slope. In other words, no surface water ponding is allowed due to variability of the pore-water pressure conditions.

Lumb, 1966 has shown that the tangent of the friction angle $(\tan f)$ conforms better to the normally distribution function than the friction angle itself. Therefore, SLOPE /W uses the tangent of the friction angles in the estimation of all trial friction angles (i.e., Phi, Phi_B and Phi_2 in a bilinear function).

NOTE: An independent normalized random number is obtained for each input parameter for each Monte Carlo trial.

Correlation Coefficient

A correlation coefficient expresses the relative strength of the association between two parameters. Laboratory tests on a wide variety of soils (Lumb, 1970; Grivas, 1981 and Wolff, 1985) show that the shear strength parameters c and f are often negatively correlated with correlation coefficient ranges from -0.72 to 0.35. Correlation between strength parameters may affect the probability distribution of a slope. SLOPE/W allows the specification of c and f correlation coefficients for all soil models using c and f parameters. Furthermore, in the case of a bilinear soil model, SLOPE/W allows the specification of correlation coefficient for f and $f2$.

Correlation coefficients will always fall between -1 and 1. When the correlation coefficient is positive, c and f are positively correlated implying that larger values of c are more likely to occur with larger values of f. Similarly, when the correlation coefficient is negative, c and f are negatively correlated and reflects the tendency of a larger value of c to occur with a smaller value of f. A zero correlation coefficient implies that c and f are independent parameters.

In SLOPE/W, when estimating a new trial value for f and $f2$, the normalized random number is adjusted to consider the effect of correlation. The following equation is used in the adjustment:

$$N_a = N_1 k + \left(1 - |k|\right) N_2 \tag{8.46}$$

where :

k = correlation coefficient between the first and second parameters
N_1 = normalized random number for the first parameter
N_2 = normalized random number for the second parameter
N_a = adjusted normalized random number for the second parameter

Statistical Analysis

SLOPE/W assumes that the trial factors of safety are normally distributed. As a result, statistical analysis can be conducted to determine the mean, standard deviation, the probability density function and the probability distribution function of the slope stability problem. The equations used in the statistical analysis are summarized as follows (Lapin, 1983):

Mean factor of safety, m :

$$m = \left(\frac{\sum_{i=0}^{n} F_i}{n} \right) \tag{8.47}$$

Standard deviation, s :

$$s = \sqrt{\left(\frac{\sum_{i=o}^{n} \left(F_i - m\right)^2}{n} \right)} \tag{8.48}$$

Probability density function:

$$f\left(F\right) = \frac{1}{s\sqrt{2p}} e^{-\frac{(F-m)^2}{2s^2}} \quad -\infty < F < \infty \tag{8.49}$$

Probability distribution function:

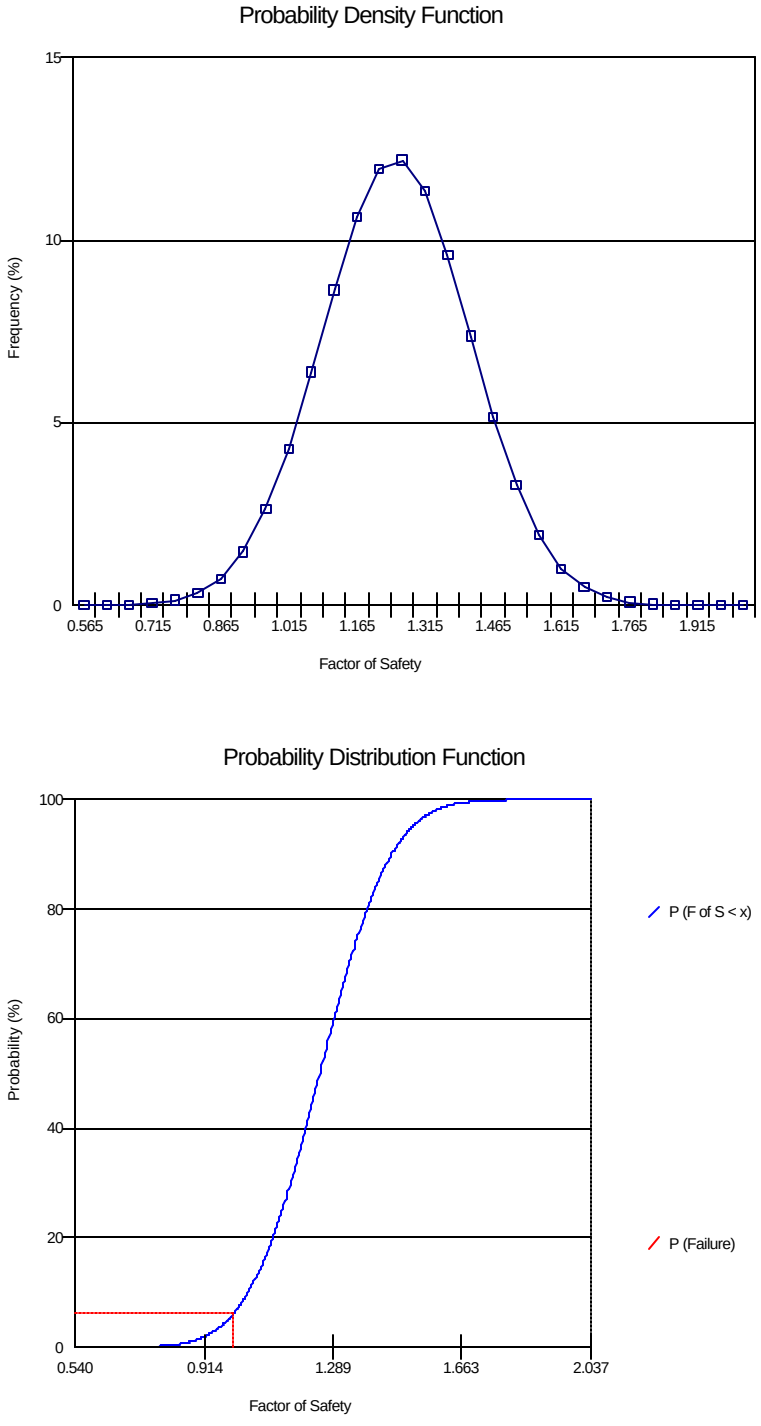
$$f\left(F\right) = P\left[X \le F\right] = \int_{-\infty}^{F} \frac{1}{s\sqrt{2p}} e^{-\frac{(F-m)^2}{2s^2}} dy \tag{8.50}$$

where:

F_i = the trial factors of safety
n = number of trial factors of safety
F = factor of safety

An example probability density function and the corresponding probability distribution function are presented in Figure 8.24.

Figure 8.24 Probability Density Function and Probability Distribution Function.



Probability of Failure and Reliability Index

A factor of safety is really an index indicating the relative stability of a slope. It does not imply the actual risk level of the slope due to the variability of input parameters. With probabilistic analysis, two useful indices are available to quantify the stability or the risk level of a slope. These two indices are known as the probability of failure and the reliability index.

As illustrated in Figure 8.24, the probability of failure is the probability of obtaining a factor of safety less than 1.0. It is computed by integrating the area under the probability density function for factors of safety less than 1.0. The probability of failure can be interpreted in two ways (Mostyn and Li, 1993):

- if a slope were to be constructed many times, what percentage of such slopes would fail, or
- the level of confidence that can be placed in a design.

The first interpretation may be relevant in projects where the same slope is constructed many times, while the second interpretation is more relevant in projects where a given design is only constructed once and it either fails or it does not. Nevertheless, the probability of failure is a good index showing the actual level of stability of a slope.

There is no direct relationship between factor of safety and probability of failure. In other words, a slope with a higher factor of safety may not be more stable than a slope with a lower factor of safety (Harr, 1987). For example, a slope with factor of safety of 1.5 and a standard deviation of 0.5 will have a much higher probability of failure than a slope with factor of safety of 1.2 and a standard deviation of 0.1.

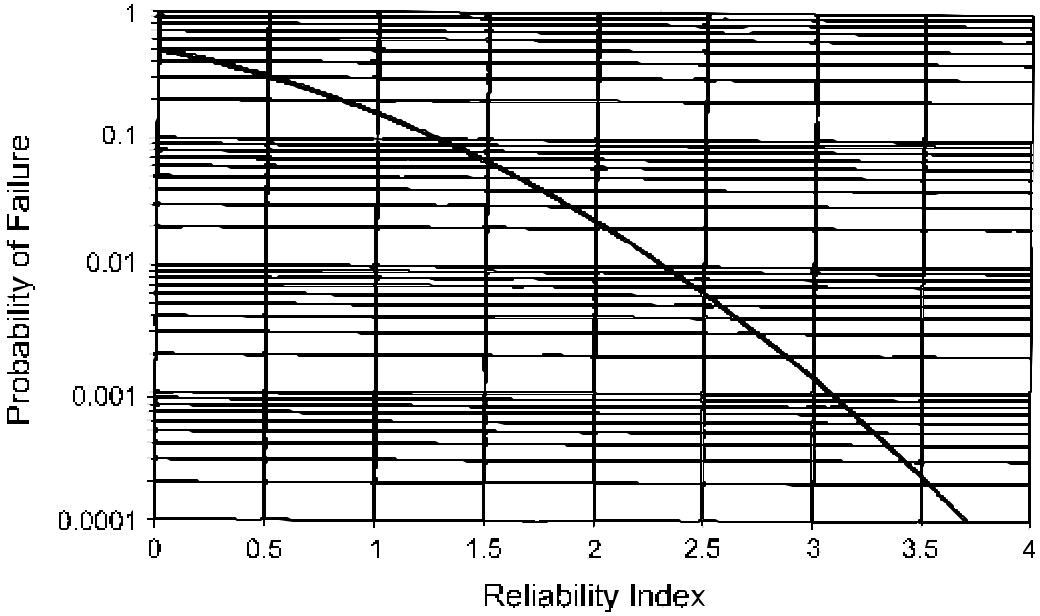
The reliability index provides a more meaningful measure of stability than the factor of safety. The reliability index (b) is defined in terms of the mean (m) and the standard deviation (s) of the trial factors of safety as (Christian, Ladd and Baecher, 1994):

$$b = \frac{(m - 1.0)}{s} \tag{8.51}$$

The reliability index describes the stability of a slope by the number of standard deviations separating the mean factor of safety from its defined failure value of 1.0. It can also be considered as a way of normalizing the factor of safety with respect to its uncertainty.

When the shape of the probability distribution is known, the reliability index can be related directly to the probability of failure. Figure 8.25 illustrates the relationship of the reliability index to the probability of failure for a normally distributed factor of safety.

Figure 8.25 Probability of Failure as a Function of the Reliability Index for a Normally Distributed Factor of Safety (After Christian, Ladd and Baecher, 1994)



Number of Monte Carlo Trials

Probabilistic slope stability analysis using the Monte Carlo method involves many trial runs. Theoretically, the more trial runs used in an analysis the more accurate the solution will be. How many trials are required in a probabilistic slope stability analysis? Harr, (1987) suggested that the number of required Monte Carlo trials is dependent on the desired level of confidence in the solution as well as the number of variables being considered. Statistically, the following equation can be developed (Harr, 1987):

$$N_{mc} = \left[\frac{\left(d^2\right)}{\left(4\left(1-e\right)^2\right)} \right]^m \tag{8.52}$$

where :

N_{mc} = number of Monte Carlo trials
e = the desired level of confidence (0 to 100%) expressed in decimal form
d = the normal standard deviate corresponding to the level of confidence
m = number of variables

The number of Monte Carlo trials increases geometrically with the level of confidence and the number of variables. For example, if the desired level of confidence is 80%, the normal standard deviate will be 1.28, the number of Monte Carlo trials will be 10 for 1 variable, 100 for 2 variables and 1,000 for 3 variables. For a 90% level of confidence, the normal standard deviate will be 1.64, the number of Monte Carlo trials will be 67 for 1 variable, 4,489 for 2 variables and 300,763 for 3 variables. In fact, for a 100% level of confidence, an infinite number of trials will be required.

For practical purposes, the number of Monte Carlo trials to be conducted is usually in the order of thousands. This may not be sufficient for a high level of confidence with multiple variables; fortunately, in most cases, the solution is not very sensitive to the number of trials after a few thousands trials have been run. Furthermore, for most engineering projects, the degree of uncertainty in the input parameters may not warrant a high level of confidence in a probabilistic analysis.

Chapter 9 Verification

Introduction **9-3**

Comparison with Hand Calculations **9-4**
Lambe and Whitman's Solution 9-4
SLOPE/W Solution Hand Calculated 9-6

Comparison with Stability Charts **9-10**
Bishop and Morgenstern's Solution 9-10
SLOPE/W Solution Stability Chart 9-10

Comparison with Closed Form Solutions **9-12**
Closed Form Solution for an Infinite Slope 9-12
SLOPE/W Solution Closed Form 9-13

Comparison Study **9-15**

Illustrative Examples **9-17**
Example with Circular Slip Surfaces 9-17
Example with Composite Slip Surfaces 9-17
Example with Fully Specified Slip Surfaces 9-18
Example with Block Slip Surfaces 9-19
Example with Pore-Water Pressure Data Points 9-20
Example with SEEP/W Pore-Water Pressure 9-21
Example with Slip Surface Projection 9-23
Example with Geofabric Reinforcement 9-23
Example with Anchors 9-25
Example with Finite Element Stresses 9-27
Example with Anisotropic Strength 9-29
Example with Probabilistic Analysis 9-33

Introduction

This chapter presents the analyses of some common problems for which there are closed form or published solutions. The purpose of presenting these analyses is:

- To provide benchmark references which can be used to verify that the software is functioning properly.
- To illustrate the use of SLOPE/W and demonstrate the capabilities of the software.

The data input files and computed output files for each problem are included with the SLOPE/W software. The files can be used to re-run each analysis and to check that the same results can be obtained as presented in this chapter.

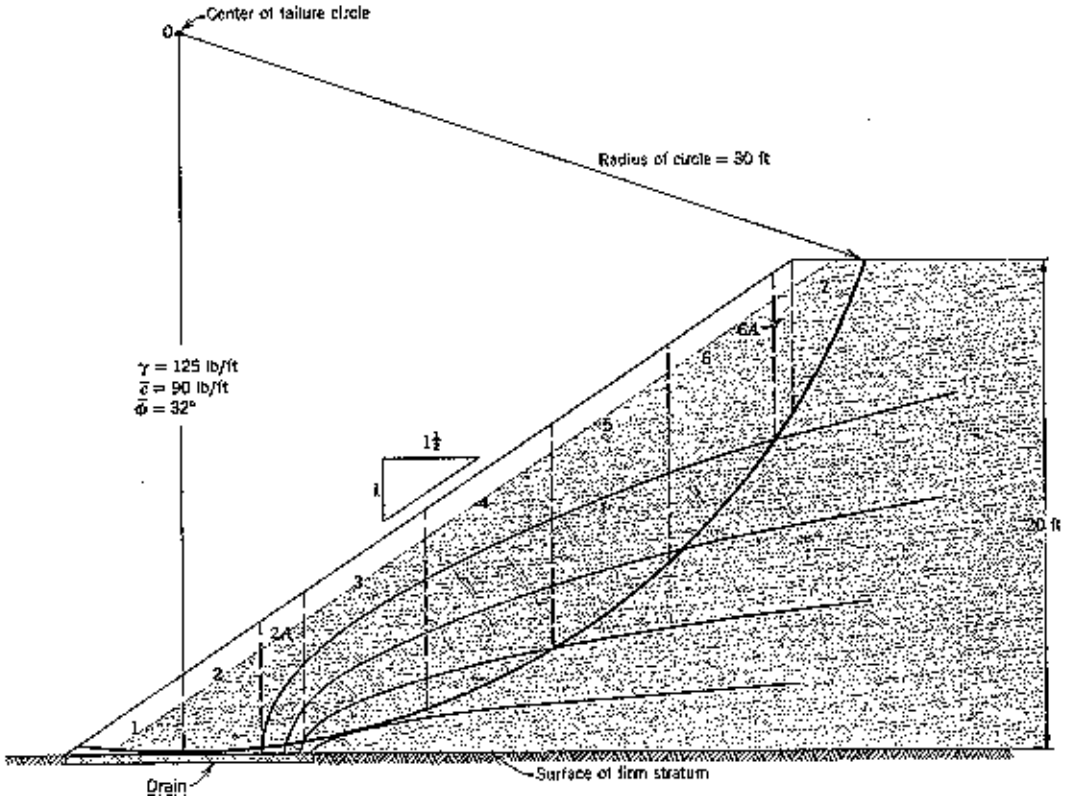
Comparison with Hand Calculations

The first verification example problem involves the comparison of SLOPE/W solutions to the hand-calculated solutions of a simple slope. The factor of safety from the Ordinary method and the Bishop's Simplified method are compared. In both cases, SLOPE/W gives the same results as the hand calculated solutions.

Lambe and Whitman's Solution

Lambe and Whitman present a hand-calculated factor of safety for a simple slope with an underdrain (Figure 9.1). The slope is 20 feet high, with a slope of 1 vertical to 1.5 horizontal. The material of the slope is homogeneous with c'= 90 psf, $f = 32°$ and $g = 125$ pcf. The slip surface is assumed to be circular with a radius of 30 feet from the center, as shown in Figure 9.1. The pore water pressure conditions for the slope are characterized by a flow net.

Figure 9.1 Stability of Slope with an Underdrain (after Lambe and Whitman)



Lambe and Whitman divide the entire sliding mass into nine slices with each slice width, average height and weight calculated as shown in Table 9.1. The total weight of the sliding mass is about 26,500 lbs.

Table 9.1 Lambe and Whitman Weight Computations

Slice	Width (ft)	Average Height (ft)	Weight (kips)
1	4.5	1.6	0.9
2	3.2	4.2	1.7
2*A*	1.8	5.8	1.3
3	5.0	7.4	4.6
4	5.0	9.0	5.6
5	5.0	9.3	5.8
6	4.4	8.4	4.6
6*A*	0.6	6.7	0.5
7	3.2	3.8	1.5
			W=2.65

Table 9.2 presents Lambe and Whitman's calculation for determining the Ordinary factor of safety. The hand-calculated factor of safety is 1.19.

Table 9.2 Lambe and Whitman Calculation of the Ordinary Factor of Safety

Slice	W_i (kips)	$\sin\alpha_i$	$W_i \sin\alpha_i$ (kips)	$\cos\alpha_i$	$W_i \cos\alpha_i$ (kips)	u_i (kips/ft)	Δl_i (ft)	U_i (kips)	$\overline{N}_i$ (kips)
1	0.9	-0.03	0	1.00	0.9	0	4.4	0	0.9
2	1.7	0.05	0.1	1.00	1.7	0	3.2	0	1.7
2A	1.3	0.14	0.2	0.99	1.3	0.03	1.9	0.05	1.25
3	4.6	0.25	1.2	0.97	4.5	0.21	5.3	1.1	3.4
4	5.6	0.42	2.3	0.91	5.1	0.29	5.6	1.6	3.5
5	5.8	0.58	3.4	0.81	4.7	0.25	6.2	1.55	3.15
6	4.6	0.74	3.4	0.67	3.1	0.11	6.7	0.7	2.4
6A	0.5	0.82	0.4	0.57	0.3	0	1.2	0	0.3
7	1.5	0.87	1.3	0.49	0.7	0	7.3	0	0.7
			12.3				41.8		17.3

$$F = \frac{0.09(41.8) + 17.3\tan 32°}{12.3} = \frac{3.76 + 10.82}{12.3} = \frac{14.58}{12.3} = 1.19$$

Lambe and Whitman also compute the Bishop's Simplified factor of safety using a trial and error approach. The computations and results are presented in Table 9.3.

Table 9.3 Lambe and Whitman Calculation of the Bishop Simplified Factor of Safety

(1) Slice	(2) Δx_i (ft)	(3) $\bar{c}\Delta x_i$ (kips)	(4) $u_i \Delta x_i$ (kips)	(5) $W_i - u_i \Delta x_i$ (kips)	(6) (5) tan ϕ (kips)	(7) (3) + (6) (kips)	(8) M_i $F = 1.25$	(8) M_i $F = 1.35$	(9) (7) ÷ (8) $F = 1.25$	(9) (7) ÷ (8) $F = 1.35$
1	4.5	0.40	0	0.9	0.55	0.95	0.97	0.97	1.0	1.0
2	3.2	0.29	0	1.7	1.05	1.35	1.02	1.02	1.3	1.3
2*A*	1.8	0.16	0.05	1.25	0.80	0.95	1.06	1.05	0.9	0.9
3	5.0	0.45	1.05	3.55	2.25	2.70	1.09	1.08	2.5	2.5
4	5.0	0.45	1.45	4.15	2.55	3.00	1.12	1.10	2.7	2.75
5	5.0	0.45	1.25	4.55	2.7	3.15	1.10	1.08	2.85	2.9
6	4.4	0.40	0.50	4.1	2.65	3.05	1.05	1.02	2.9	2.95
6*A*	0.6	0.05	0	0.5	0.30	0.35	0.98	0.95	0.35	0.4
7	3.2	0.29	0	1.5	0.95	1.25	0.93	0.92	1.3	1.35
									15.8	16.05

For assumed $F = 1.25$ $\quad F = \dfrac{15.8}{12.3} = 1.29$

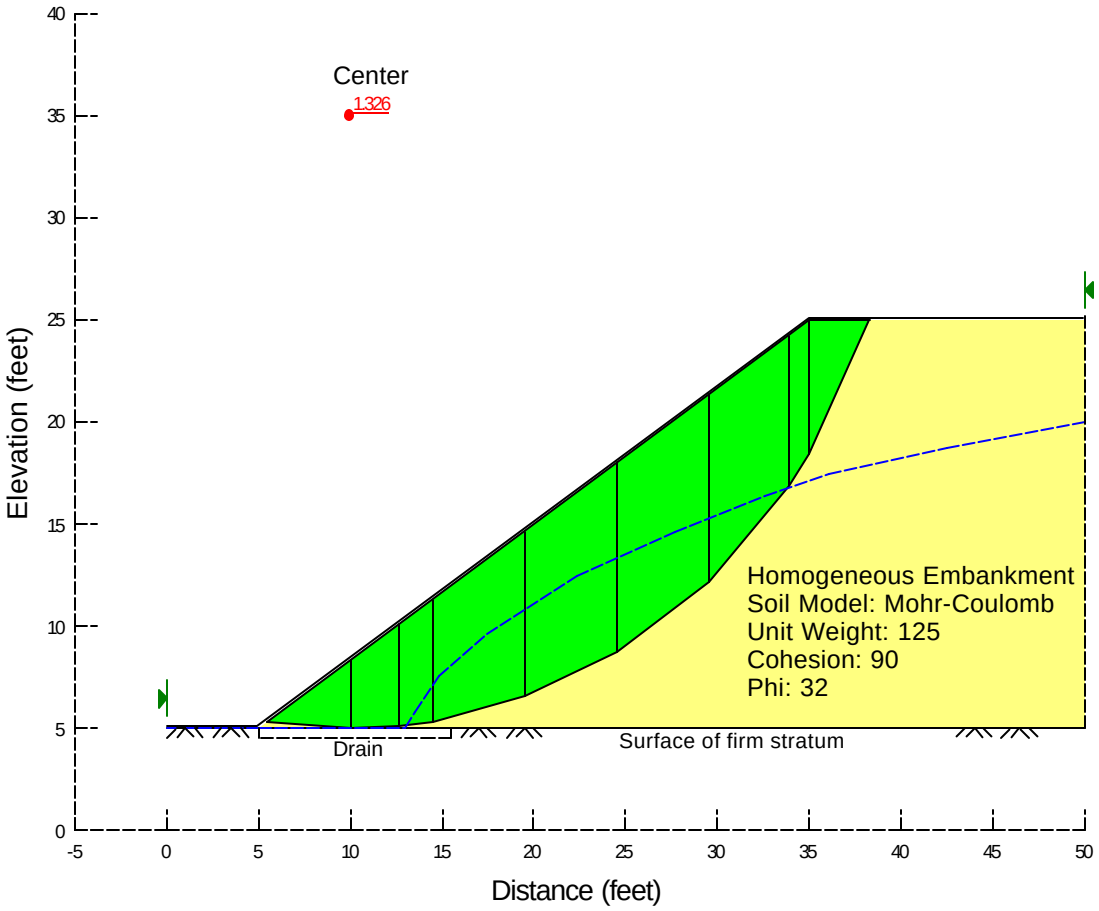
$F = 1.35$ $\quad F = \dfrac{16.05}{12.3} = 1.31$

As shown in the above calculations, a trial factor of safety of 1.25 results in a computed factor of safety of 1.29, and a trial factor of safety of 1.35 results in a computed value of 1.31. Since the trial value of 1.25 is too low and the trial value of 1.35 is too high, the correct value using the Bishop Simplified method is between 1.25 and 1.35.

SLOPE/W Solution Hand Calculated

The same problem is analyzed using SLOPE/W. The associated files are named LAM-WHIT. Figure 9.2 shows the same slope as modelled by SLOPE/W.

Figure 9.2 Stability of Slope with an Underdrain Using SLOPE/W



Nine slices are also used in the analysis. The computed slice width, average height and weight of the sliding mass is tabulated in Table 9.4. The total weight of the sliding mass is 26,714 lbs. The slices modelled by SLOPE/W are essentially the same as those used by Lambe and Whitman.

Table 9.4 SLOPE/W Weight Computations

Slice #	Width	Average Height	Weight
1	4.5	1.75	984.4
2	2.7	4.20	1417.5
3	1.8	5.52	1242.0
4	5.0	7.17	4481.3
5	5.0	8.83	5518.8
6	5.0	9.39	5868.8
7	4.4	8.52	4686.0
8	1.1	7.02	965.3
9	3.3	3.79	1563.4
			26727.5

Table 9.5 presents the hand calculated factor of safety for the Ordinary method using the slice quantities computed by SLOPE/W. The slice quantities in Tables 9.4 and 9.5 are taken from the forces file LAM-WHIT.FRC and can be displayed using the View Slice Forces command in SLOPE/W CONTOUR.

Table 9.5 SLOPE/W Calculation of the Ordinary Factor of Safety

Slice #	Weight	α	Weight x $\sin\alpha$	Normal	Water	Normal–Water	Base Length
1	984.4	-4.3	-73.8	980.9	0.0	980.9	4.51
2	1417.5	2.58	63.8	1417.1	0.0	1417.1	2.70
3	1242.0	6.90	149.2	1232.2	81.6	1150.6	1.81
4	4481.3	13.5	1046.2	4357.8	1110.5	3247.3	5.14
5	5518.8	23.7	2218.3	5053.3	1607.9	3445.4	5.46
6	5868.8	34.7	3341.0	4821.1	1529.0	3292.1	6.08
7	4686.0	46.70	3410.3	3213.2	699.6	2513.6	6.41
8	965.3	54.6	786.8	558.5	0.0	558.5	1.90
9	1563.4	63.50	1399.1	694.8	0.0	694.8	7.36
Total	26727.5		12340.9			17300.3	41.37

From the total values in Table 9.4, the Ordinary factor of safety can be calculated as:

$$\text{Factor of Safety} = \frac{90\,psf \times 41.37 + 17300.3 \tan 32}{12340.9} = 1.18$$

The SLOPE/W computed value as listed in the LAM-WHIT.FAC file is 1.185. Except for a slight difference due to rounding errors, SLOPE/W gives the same factor of safety as the Lambe and Whitman hand calculated value of 1.19.

The Bishop Simplified factor of safety computed by SLOPE/W is 1.326, which is within the range calculated by Lambe and Whitman (Table 9.3). This value is shown in Figure 9.2 and is recorded in the LAM-WHIT.FAC file.

Comparison with Stability Charts

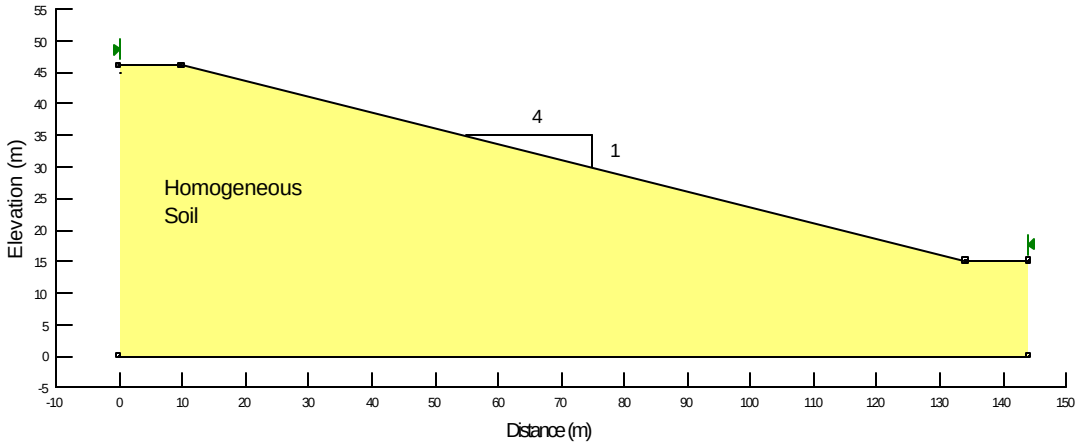
The second verification example problem is to compare the SLOPE/W solution with a factor of safety estimated using stability charts.

Bishop and Morgenstern's Solution

Bishop and Morgenstern, 1960, developed a series of stability charts that can be used to estimate the factors of safety for simple homogeneous earth slopes.

Figure 9.3 shows a 4:1 slope with c' = 12.5 kN/m^2, ϕ' = 20°, γ = 16 kN/m^3, and $r_u = 0.35$.

Figure 9.3 Homogeneous Slope Example



The height of the slope from crest to toe is 31 m. Therefore, the dimensionless parameter is,

$$\frac{c'}{\gamma H} = \frac{12.5}{16 \times 31} = 0.025$$

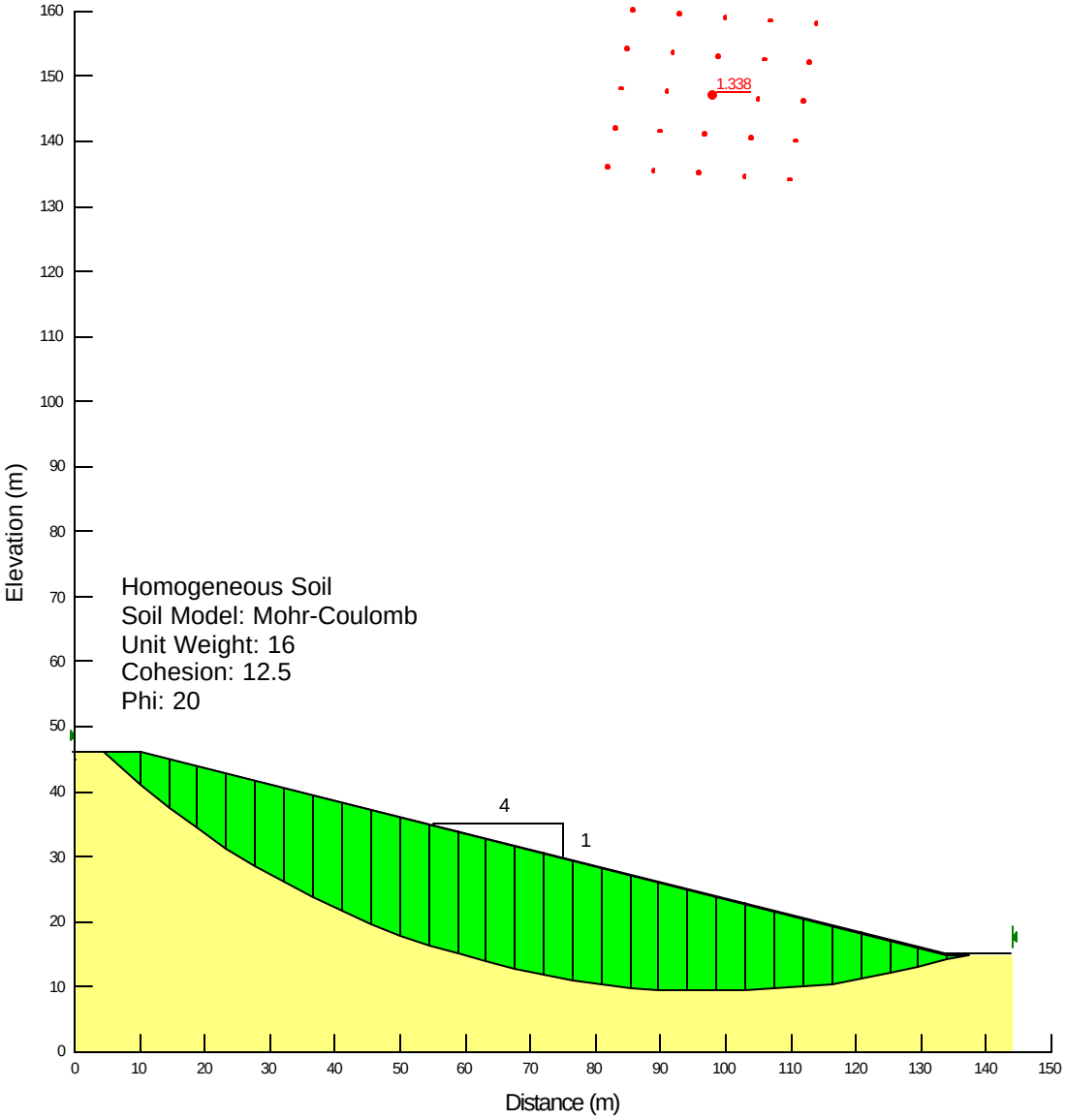
The appropriate D factor for $r_u = 0.35$ is 1.25. The stability coefficients from the Bishop and Morgenstern charts are m = 1.97 and n = 1.78. Therefore, the factor of safety of the slope using stability charts is calculated to be 1.35 as shown below:

$$\text{Factor of Safety} = m - n\,r_u = 1.97 - 1.78 \times 0.35 = 1.35$$

SLOPE/W Solution Stability Chart

The same slope is analyzed with SLOPE/W. The sliding mass contains 30 slices. The associated files for this example are named CHART. Figure 9.4 illustrates the slope modelled with SLOPE/W.

Figure 9.4 Critical Slip Surface of Homogeneous Slope



The factor of safety computed by SLOPE/W is 1.336 for both the Bishop Simplified method and the Morgenstern-Price method. This is in essence the same value as presented by the stability charts.

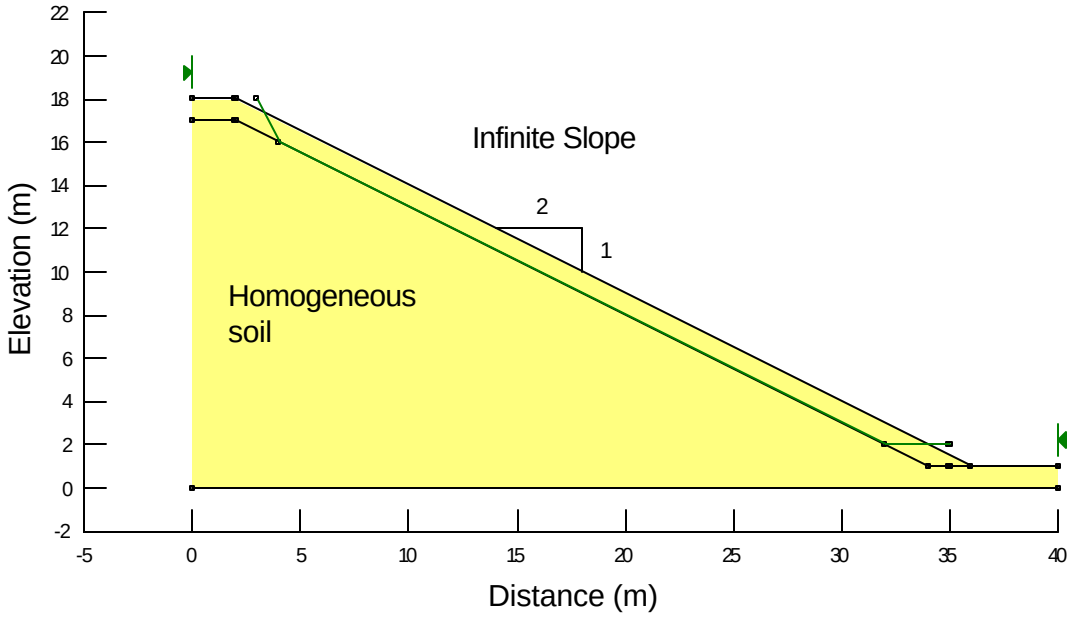
Comparison with Closed Form Solutions

The third verification example problem is to compare the factors of safety computed by SLOPE/W with closed form solutions. Three cases of an infinite slope are considered. In all cases, the factors of safety computed by SLOPE/W are identical to the closed form solutions.

Closed Form Solution for an Infinite Slope

Consider the case of a 2:1 slope, as illustrated in Figure 9.5. The sliding mass is assumed to be parallel to the slope surface (infinite slope).

Figure 9.5 Homogeneous Infinite Slope



Three cases of the closed form solutions are considered. The factors of safety for the three cases are calculated assuming the parameters shown in Table 9.6.

Table 9.6 Parameter Values Used for Infinite Slope Analyses

Parameter	Values Used		
	Case 1	**Case 2**	**Case 3**
frictional angle f', degrees	35.0°	35.0°	35.0°
cohesion *c*', kPa	0	0	5.0
pore-water coefficient r_u	0	0.25	0.25
unit weight g, kN/m^3	19.62	19.62	19.62
steepness a, degrees	26.565 (2:1 slope)	26.565 (2:1 slope)	26.565 (2:1 slope)
vertical height *H*, m	1.0	1.0	1.0

Case 1: Dry Frictional Material with No Cohesion

For a dry infinite slope consisting of a frictional material with no cohesion, the factor of safety is,

$$\text{Factor of Safety} = \frac{\tan f'}{\tan a} = \frac{\tan 35.0}{\tan 26.565} = 1.400 \tag{9.1}$$

Case 2: Wet Frictional Material with No Cohesion

For a frictional material with no cohesion under the conditions of flow parallel to the slope (i.e., pore-water pressure characterized by r_u), the factor of safety is,

$$\begin{aligned}\text{Factor of Safety} &= \left(1 - r_u \sec^2 a\right) \frac{\tan f'}{\tan a} \\ &= (1 - 0.25 * \sec^2 26.565) * 1.400 \\ &= 0.963\end{aligned} \tag{9.2}$$

Case 3: Wet Frictional Material with Cohesion

For a frictional material with cohesion under the conditions of flow parallel to the slope, the factor of safety is,

$$\begin{aligned}\text{Factor of Safety} &= \frac{c'}{g\, H \cos a \sin a} + \left(1 - r_u \sec^2 a\right) \frac{\tan f'}{\tan a} \\ &= \frac{5.0}{19.62 * 1.0 * \cos 26.565 * \sin 26.565} + 0.963 \\ &= 1.600\end{aligned} \tag{9.3}$$

SLOPE/W Solution Closed Form

The three cases of the infinite slope are analyzed using SLOPE/W. Figure 9.6 illustrates the solution for Case 1, where the Morgenstern-Price method is used. The sliding mass is simulated with 30 slices.

The associated SLOPE/W files for Case 1 are named INFINITE. The data file for Case 2 can be obtained by changing the pore-water coefficient (r_u) value of the soil in Case 1 from 0.0 to 0.25. Case 3 can be obtained by changing the cohesion (c) value in Case 2 from 0.0 to 5.0.

Figure 9.6 SLOPE/W Solutions of the Infinite Slope

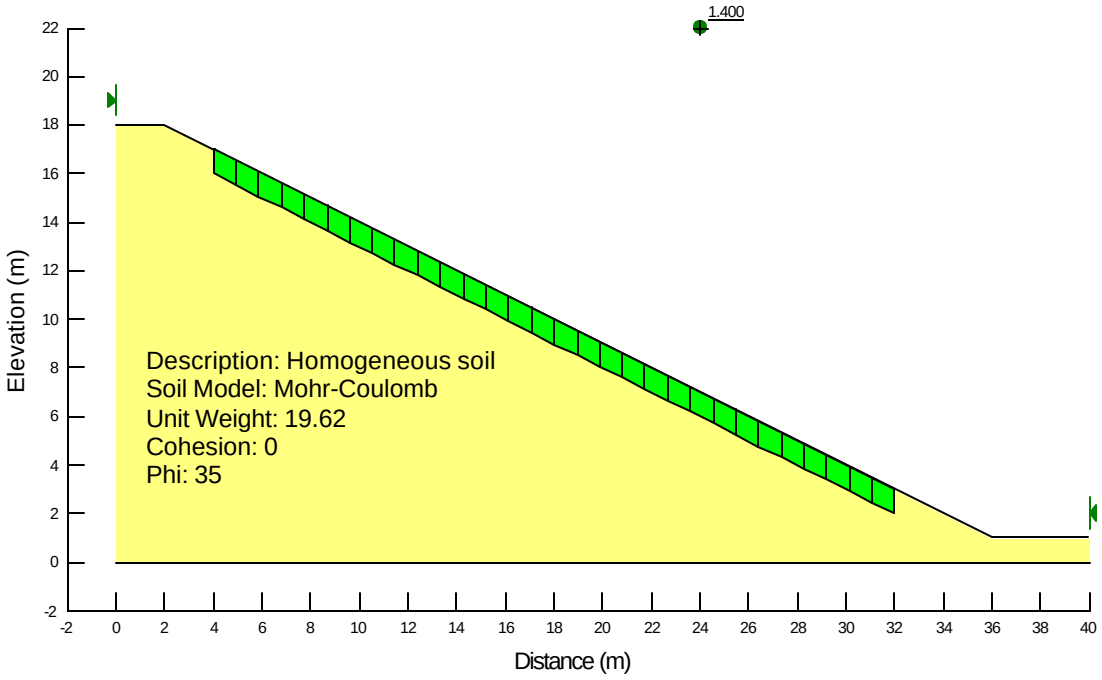


Table 9.7 tabulates the comparison of the three cases between the closed form solutions and the SLOPE/W solutions. Factors of safety for the Bishop Simplified method and the Morgenstern-Price method are presented. In all cases, SLOPE/W gives essentially the same factors of safety as the closed form solutions.

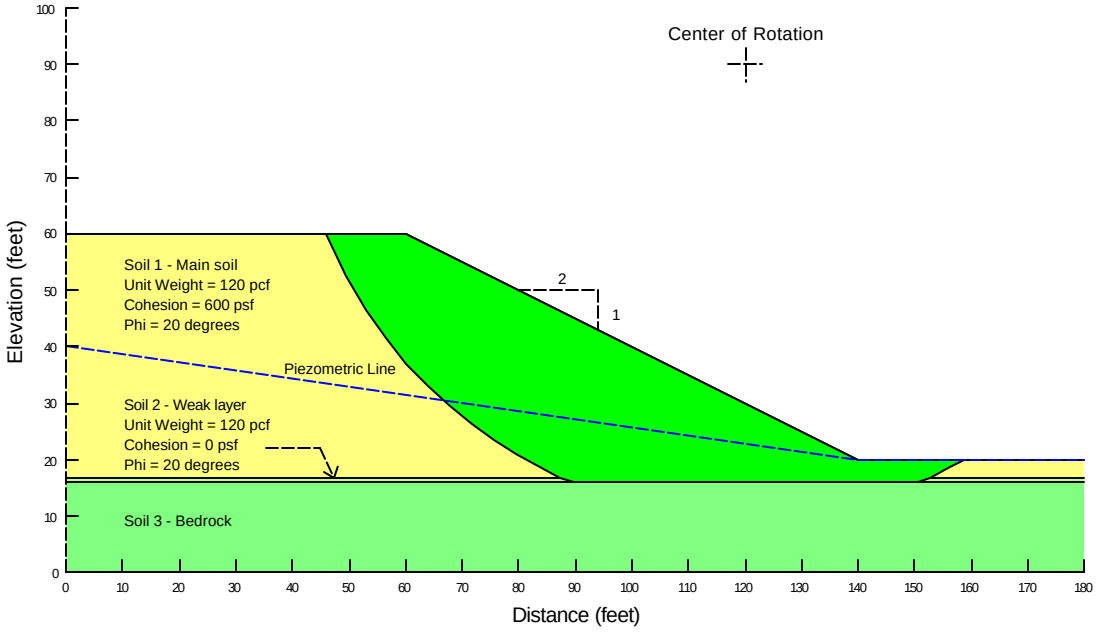
Table 9.7 Comparison of SLOPE/W Solutions with Closed Form Solutions for an Infinite Slope

Case	f'	c'	r_u	Factor of Safety		
				Closed Form Solution	SLOPE/W Bishop Simplified	SLOPE/W Morgenstern-Price
1	35	0.0	0.0	1.400	1.402	1.400
2	35	0.0	0.25	0.963	0.965	0.963
3	35	5.0	0.25	1.600	1.601	1.600

Comparison Study

Fredlund and Krahn, 1977, analyzed the problem presented in Figure 9.7 as part of a comparison study of slope stability methods. They used the original Morgenstern-Price computer program, as modified at the University of Alberta, (Krahn, Price and Morgenstern, 1971), and compared it with a slope stability program developed by Fredlund, 1974, at the University of Saskatchewan.

Figure 9.7 Example Problem for Comparison of Different Computer Programs



The problem shown in Figure 9.7 was reanalyzed with SLOPE/W. The soil properties and the pore-water pressure conditions of the slope were varied to simulated several cases of the slope. Table 9.8 presents the factors of safety and lambda values as computed by the University of Alberta, University of Saskatchewan, and SLOPE/W computer programs with a constant side force function.

The associated SLOPE/W files for Case 6 (with piezometric line, weak layer, and bedrock) are named COMPARE. The data file for Case 1 can be obtained by making all soil properties the same as Soil 1 and selecting no pore-water pressure. The data file for Case 2 can be obtained by adding the soil properties of the weak layer (Soil 2) and the soil properties of the bedrock (Soil 3) to Case 1. The data file for Case 3 can be obtained by changing the pore-water coefficient (r_u) value of the soil in Case 1 from 0.0 to 0.25. The data file for Case 4 can be obtained by changing the pore-water coefficient (r_u) value of the soil in Case 2 from 0.0 to 0.25. The data file for Case 5 can be obtained by adding the piezometric line to Soils 1 to 3 in Case 1.

Table 9.8 Comparison of SLOPE/W with Other Computer Programs

		U. of A.		U. of S.		SLOPE/W	
Case	**Description**	**F of S**	λ	**F of S**	λ	**F of S**	λ
1	No pore-water pressure, no weak layer, no bedrock	2.085	0.257	2.076	0.254	2.071	0.262
2	No pore-water pressure, with weak layer & bedrock	1.394	0.182	1.378	0.159	1.338	0.182
3	With r_u = 0.25, no weak layer, no bedrock	1.772	0.351	1.765	0.244	1.756	0.253
4	With r_u = 0.25, with weak layer & bedrock	1.137	0.334	1.124	0.116	1.081	0.157
5	With piezometric line, no weak layer, no bedrock	1.838	0.270	1.833	0.234	1.827	0.245
6	With piezometric line, with weak layer & bedrock	1.265	0.159	1.250	0.097	1.212	0.130

SLOPE/W gives essentially the same factors of safety as the University of Alberta and University of Saskatchewan computer programs. The small differences are principally due to slight differences in geometric interpretation of the sections and different procedures for subdividing the potential sliding mass into slices.

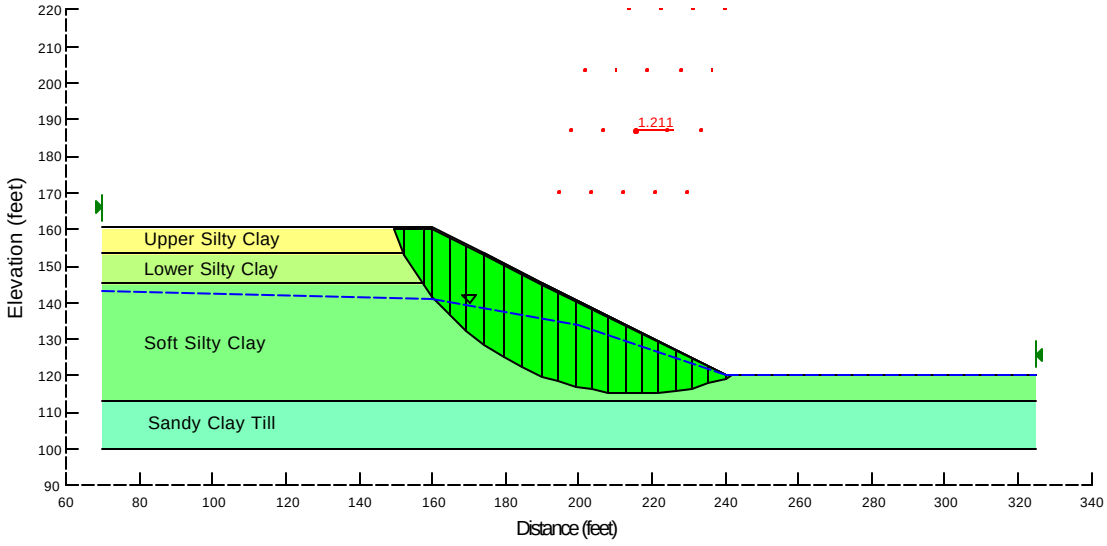
Illustrative Examples

Ten illustrative examples are included in the SLOPE/W software package to show the various types of problems that can be analyzed with SLOPE/W. The file names, the primary purpose, the analysis method, and the special features of each example problem are listed in the following sections.

Example with Circular Slip Surfaces

- File Name: CIRCLE
- Primary Purpose: To show how to analyze a case with circular slip surfaces.
- Analysis Method: Bishop simplified method
- Special Features: Circular slip surfaces, 20 points in search grid, multiple soil layers, and pore-water pressure specified by a piezometric line.

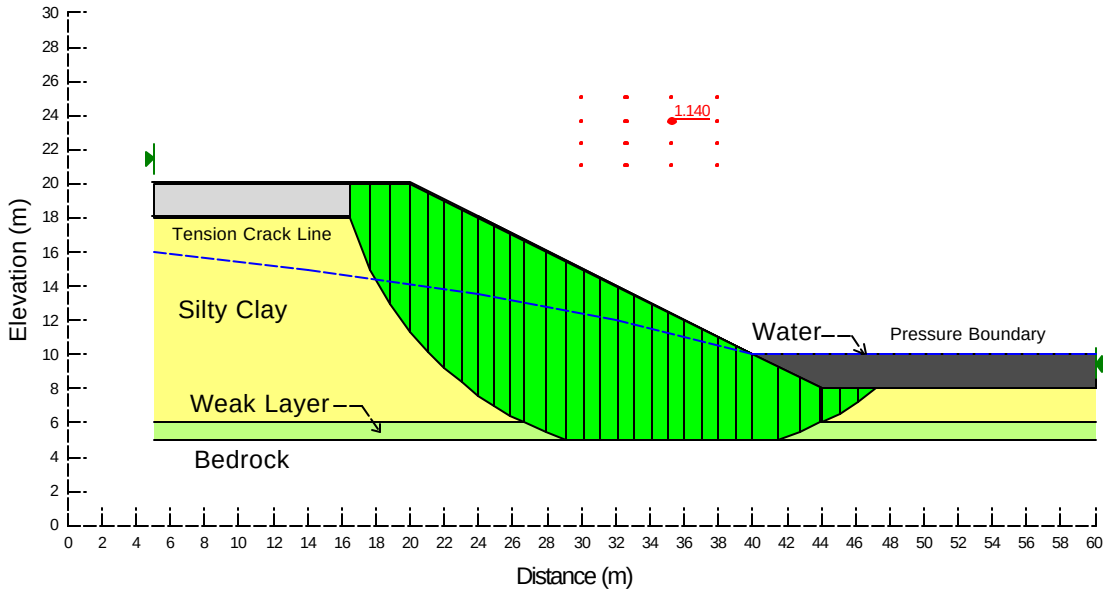
Figure 9.8 Example 1 - CIRCLE



Example with Composite Slip Surfaces

- File Name: COMPOSIT
- Primary Purpose: To illustrate the use of composite slip surfaces.
- Analysis Method: Morgenstern-Price method.
- Special Features: Composite slip surfaces, tension crack specified by a tension crack line, pore-water pressure specified by a piezometric line, and downstream water ponding simulated by a pressure line boundary.

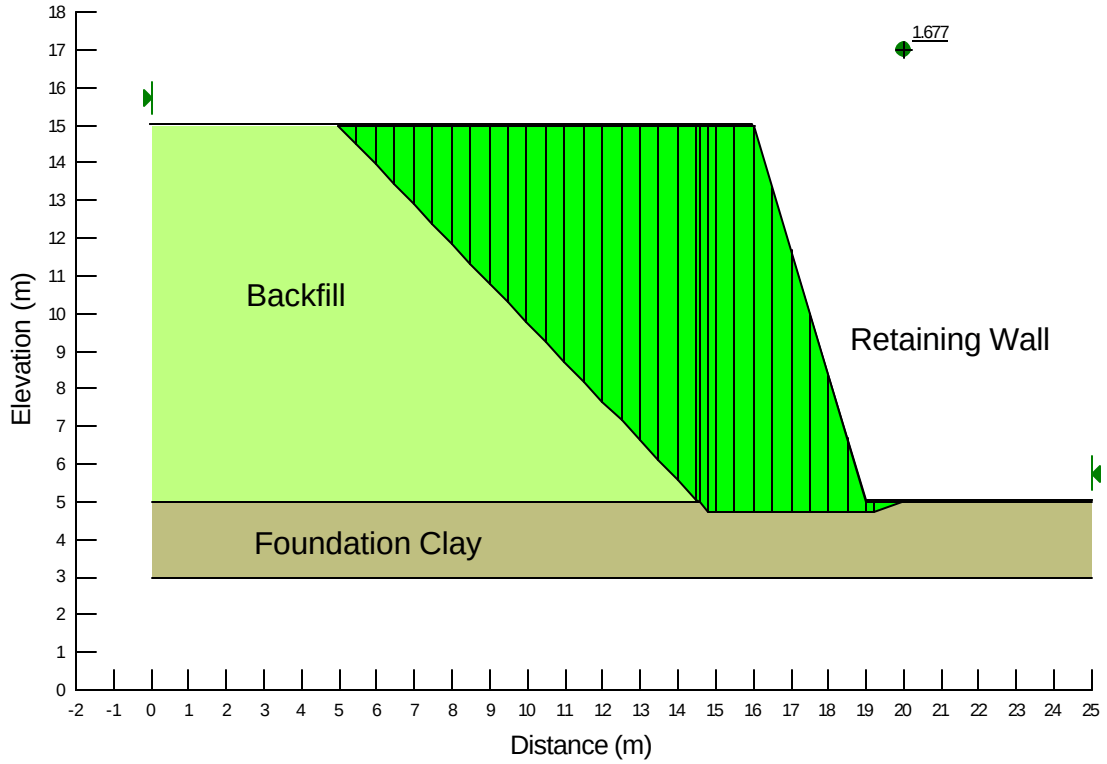
Figure 9.9 Example 2 - COMPOSIT



Example with Fully Specified Slip Surfaces

- File Name: SPECIFY
- Primary Purpose: To suggest a procedure for analyzing the stability of a gravity retaining wall using fully specified slip surfaces.
- Analysis Method: Spencer method.
- Special Features: Fully specified slip surfaces, a single search center, retaining wall, and no pore-water pressure.

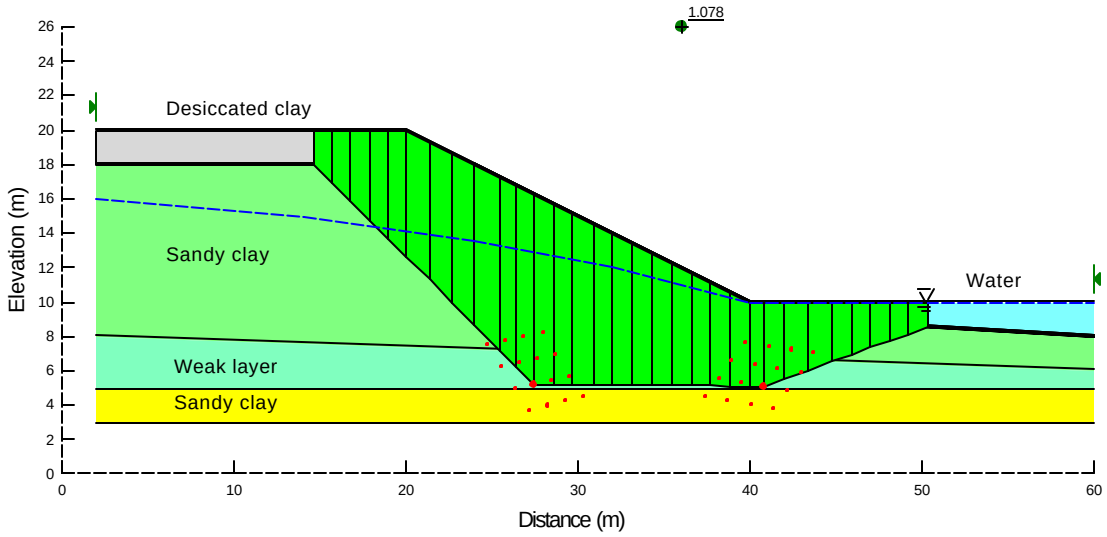
Figure 9.10 Example 3 - SPECIFY



Example with Block Slip Surfaces

- File Name: BLOCK
- Primary Purpose: To illustrate the use of the block search technique to generate a series of slip surfaces.
- Analysis Method: Morgenstern-Price method.
- Special Features: Block specified slip surfaces, tension crack specified by a tension crack line, and downstream water ponding simulated with a no strength soil layer.

Figure 9.11 Example 4 - BLOCK



Example with Pore-Water Pressure Data Points

- File Name: PNT-PWP
- Primary Purpose: To present a case with pore-water pressure head specified at discrete points.
- Analysis Method: GLE method.
- Special Features: A single circular slip surface, tension crack specified by a limiting angle, pore-water pressure specified by pressure head at discrete points, and a plot showing factor of safety versus lambda values (Figure 9.13).

Figure 9.12 Example 5 - PNT-PWP

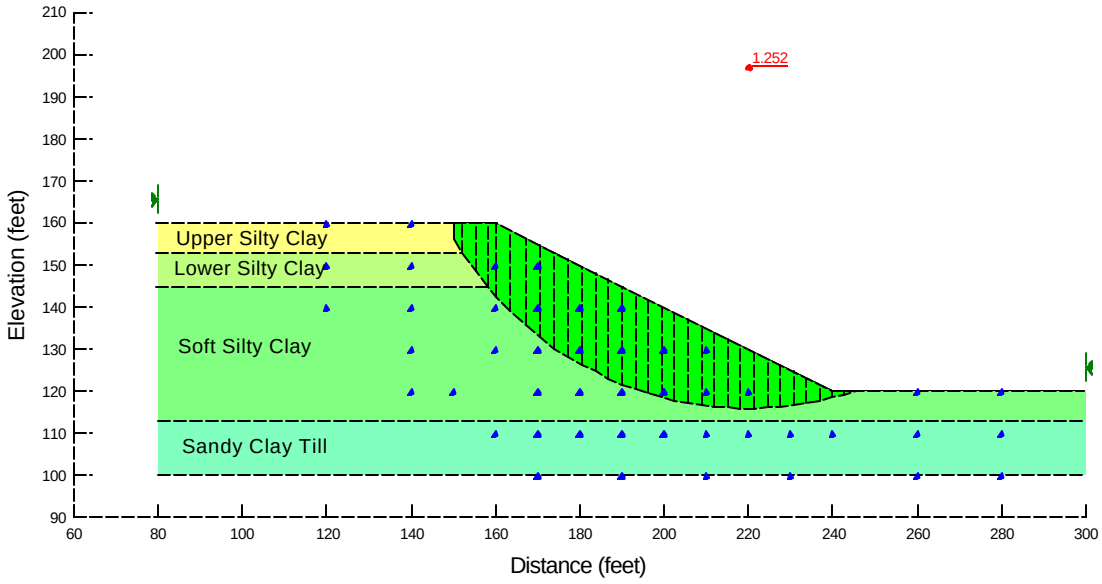
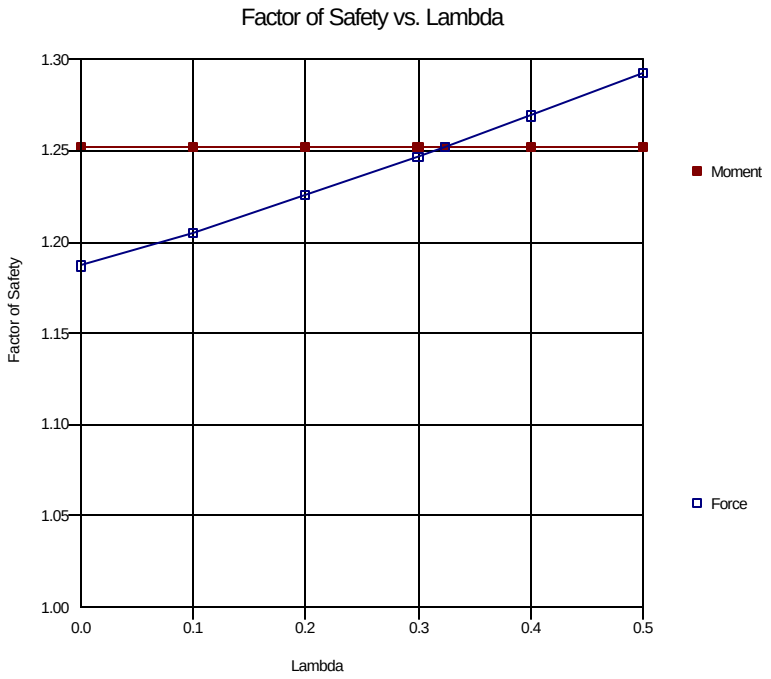


Figure 9.13 Factor of Safety as a Function of Lambda Values



Example with SEEP/W Pore-Water Pressure

- File Name: SEEP-PWP
- Primary Purpose: To illustrate the use of finite element computed pore-water pressure conditions.
- Analysis Method: Spencer method.
- Special Features: Circular slip surfaces, 25 points in search grid, pore-water pressure specified by total head computed from SEEP/W, (SEEP-PWP.H00), and increase in shear strength due to matric suction.

Figure 9.14 Example 6 - SEEP-PWP

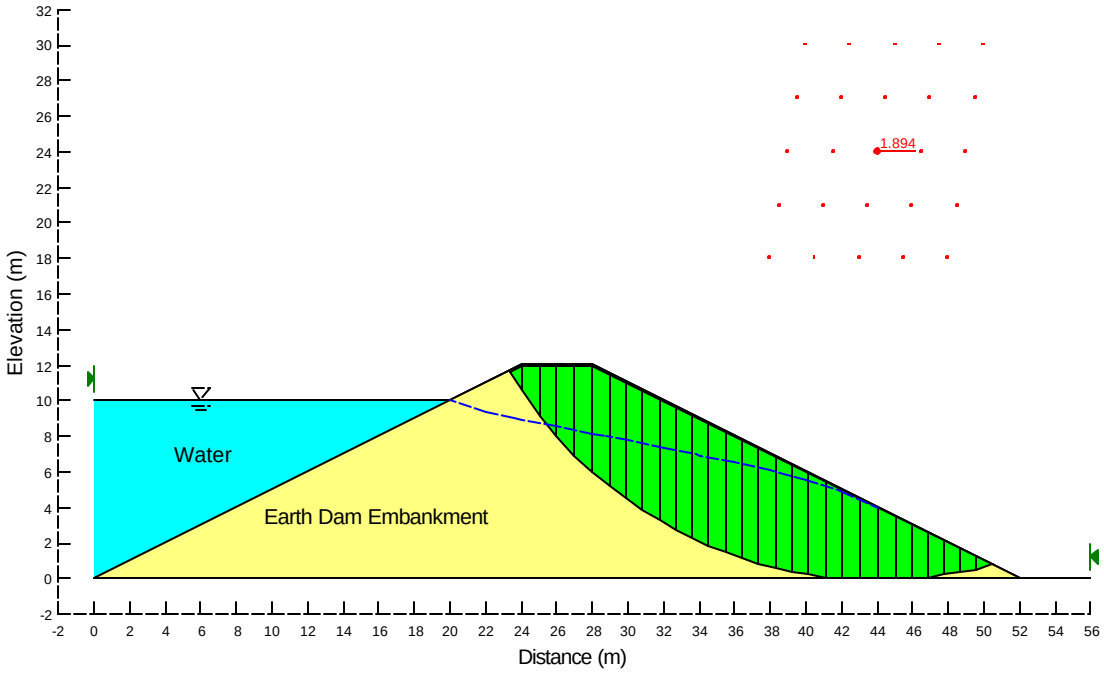


Figure 9.15 Shear Strength Components versus Slice #

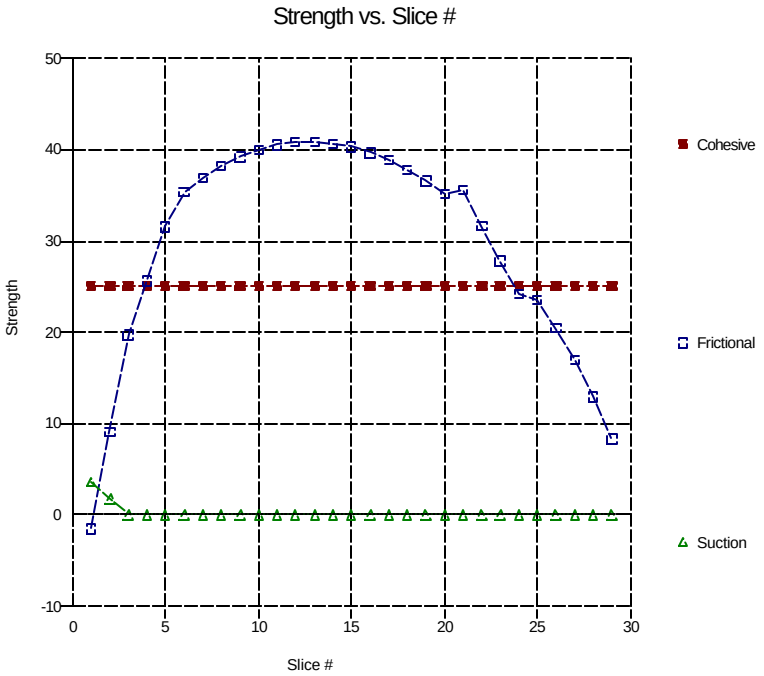
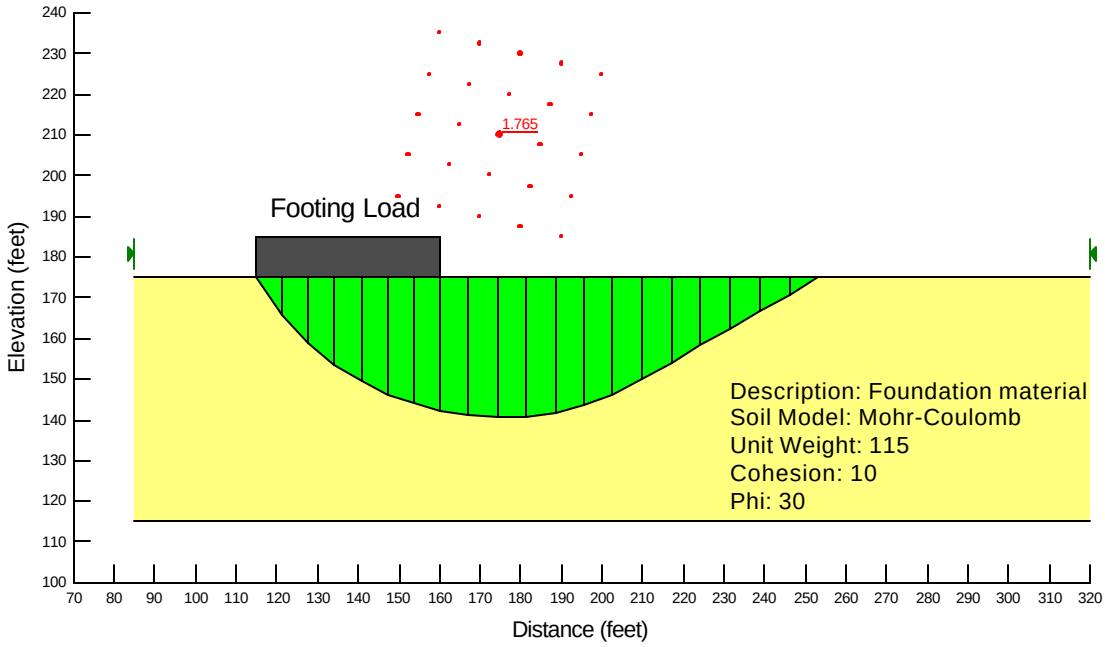


Figure 9.15 shows the distribution of the strength components along the slip surface. Note the contribution arising from the negative pore-water pressure above the piezometric line. This suction strength is included in the graph, since a f^b value is specified for the soil.

Example with Slip Surface Projection

- File Name: FOOTING
- Primary Purpose: To illustrate the use of the slip surface projection feature to simulate slip surface under a footing load.
- Analysis Method: Spencer method
- Special Features: Circular slip surfaces with a specified projection angle, radius of slip surface fixed at the lower left corner of the footing, horizontal ground surface, footing load simulated by pressure line boundary, and pore-water pressure specified by r_u coefficients.

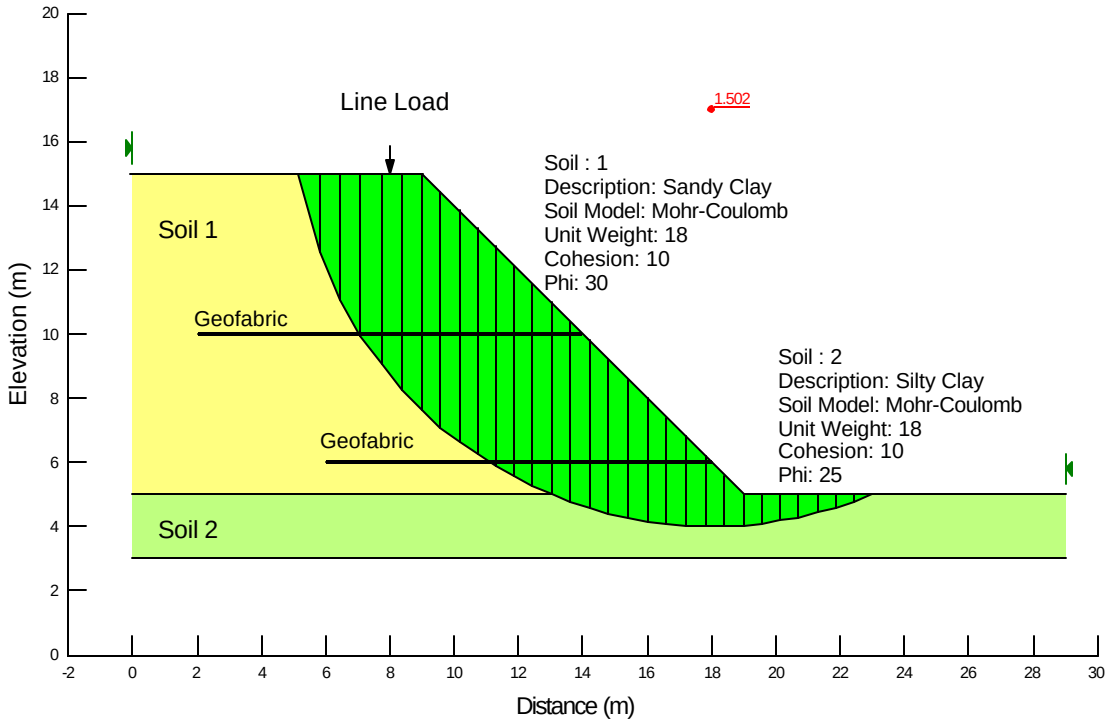
Figure 9.16 Example 7 - FOOTING



Example with Geofabric Reinforcement

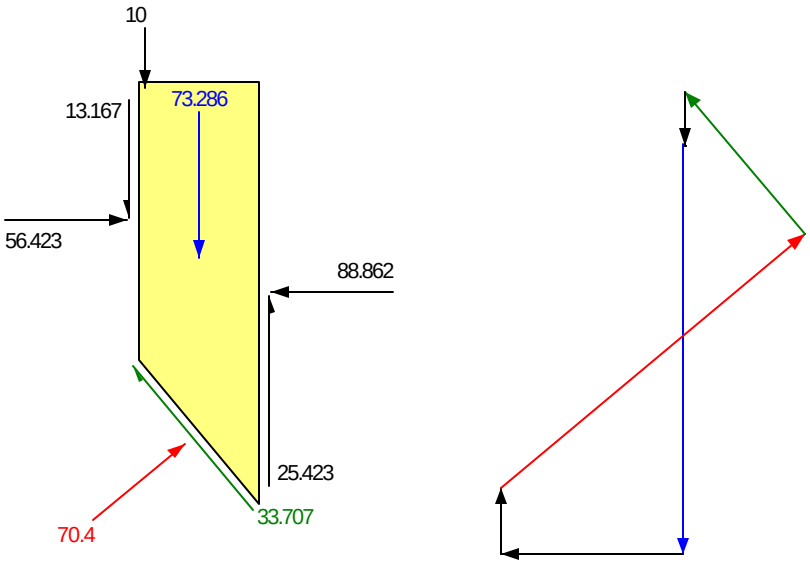
- File Name: FABRIC
- Primary Purpose: To show how geofabric reinforcement can be simulated in an analysis.
- Analysis Method: GLE method.
- Special features: A single circular slip surface, applied line load on crest of slope, geofabric reinforcements simulated as anchor loads with full bonded, length and a variable applied load, and no pore water pressure.

Figure 9.17 Example 8 - FABRIC



The free body diagrams and force polygons for Slice 5 and Slice 10 are shown in Figures 9.18 and 9.19, respectively. The line load applied on the crest of the slope is specified as 10 kN, which is shown correctly on the free body diagram in Figure 9.18.

Figure 9.18 Free Body Diagram and Force Polygon of Slice 5



The horizontal force acting on the base of Slice 10 due to the geofabric reinforcement is computed based on the specified total capacity of the geofabric, the specified bonded length and the calculated effective length behind the slip surface. For the geofabric reinforcement in Slice 10, the total specified total capacity is 50 kN, the specified bonded length is 12 m, and the calculated effective length behind the slip surface is 5.08 m. Therefore, the applied force at the base of Slice 10 due to the geofabric can be calculated as:

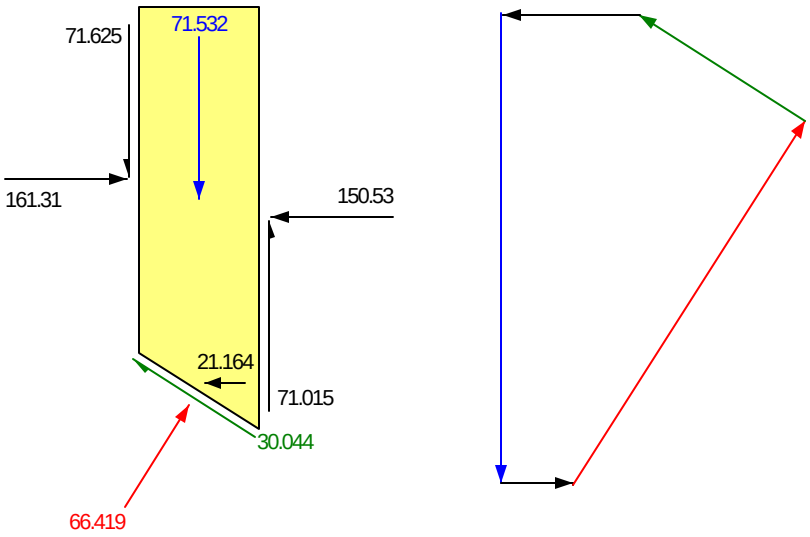
$$\text{Applied Force} = \frac{\text{effective length}}{\text{bonded length}} \times \text{specified total capacity}$$

$$\text{Applied Force} = \left(\frac{5.08\ \text{m}}{12.0\ \text{m}}\right) \times 50\ \text{kN}$$

$$= 21.2\ \text{kN}$$

This is the same as the applied force (21.164 kN) shown on the free body diagram for Slice 10.

Figure 9.19 Free Body Diagram and Force Polygon for Slice 10

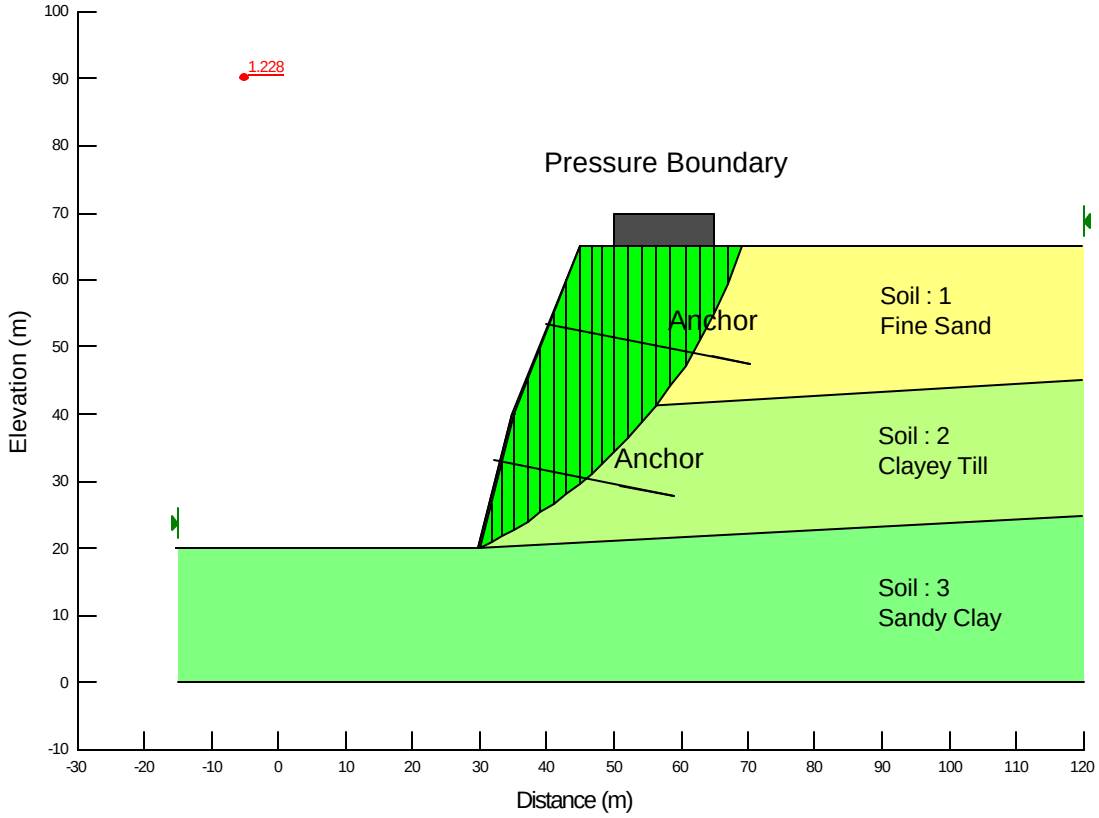


Example with Anchors

- File Name: ANCHOR
- Primary Purpose: To present a case with anchor loads.
- Analysis Method: Morgenstern-Price method.

- Special Features: A single circular slip surface with radius fixed at the toe of the slope, direction of movement from right to left, no pore water pressure, surcharge load simulated as a pressure line boundary, and anchor loads with partial bonded length and constant applied loads.

Figure 9.20 Example 9 - ANCHOR

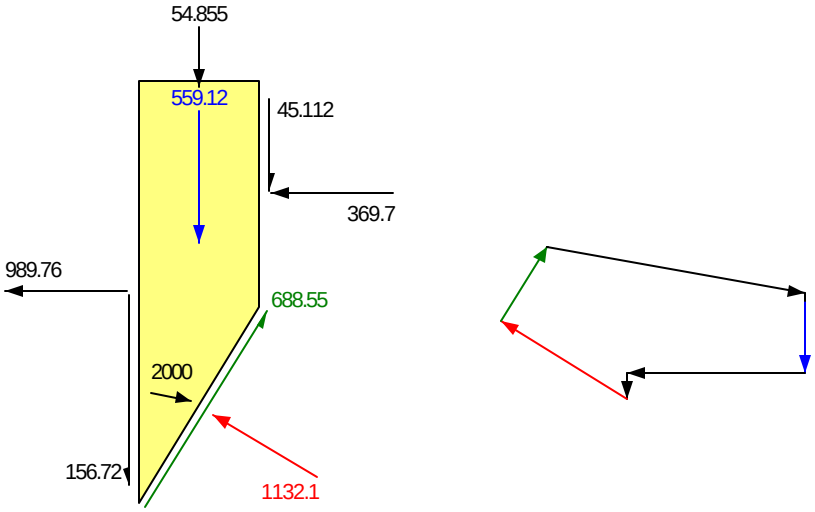


The free body diagram and force polygon for Slice 17 is shown in Figure 9.21. The specified anchor load is 2000 kN, which is shown correctly on the free body diagram of Slice 17. The applied surcharge is simulated with a pressure line boundary of 5 kN per m height per m width of slice. Since the pressure line is 5 m above the top of the slice and the width of slice is 2.1942 m, the applied surcharge load on Slice 17 can be calculated as:

Applied surcharge load = 5 kN/m/m * 5 m * 2.1942 m = 54.855 kN

This is identical to the computed force used by SLOPE/W as shown on the free body diagram.

Figure 9.21 Free Body Diagram and Force Polygon for Slice 17



Example with Finite Element Stresses

- File Name: FEM
- Primary Purpose: To illustrate the use of finite element stress for stability analysis.
- Analysis Method: Finite Element Stress method.
- Special Features: Circular slip surfaces with a single search center, finite element stress computed from SIGMA/W (FEM.S01), pore-water pressure specified by total head computed from SEEP/W, (FEM.H00), and local stability factor along the sliding mass (Figure 9.23).

Figure 9.22 Example 10 - FEM

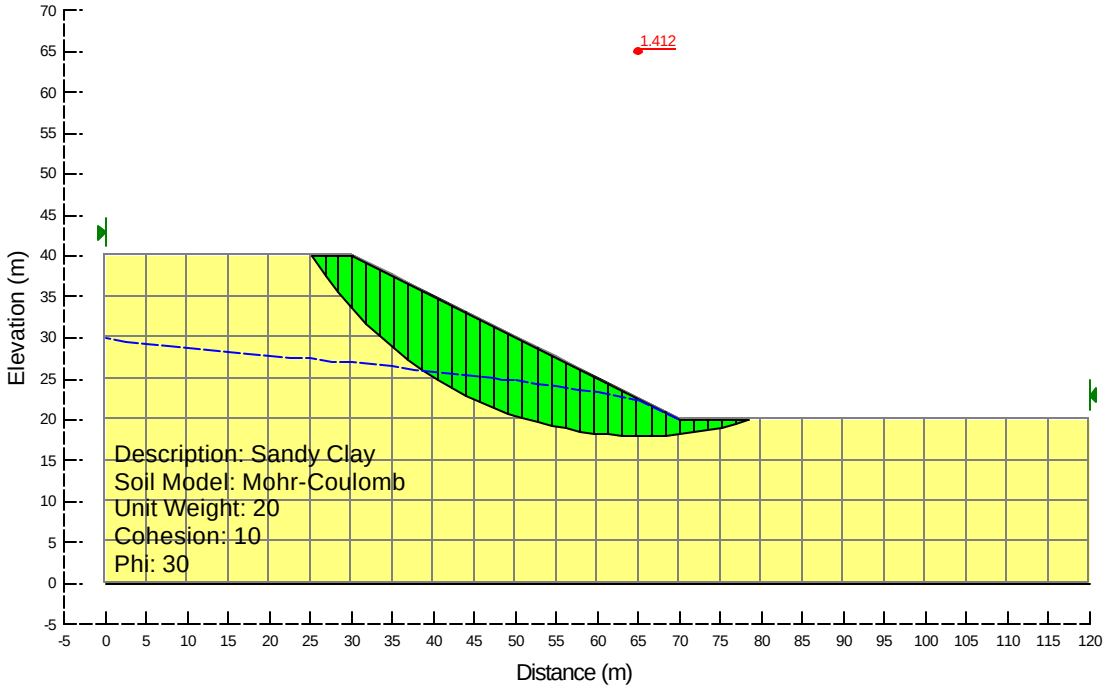
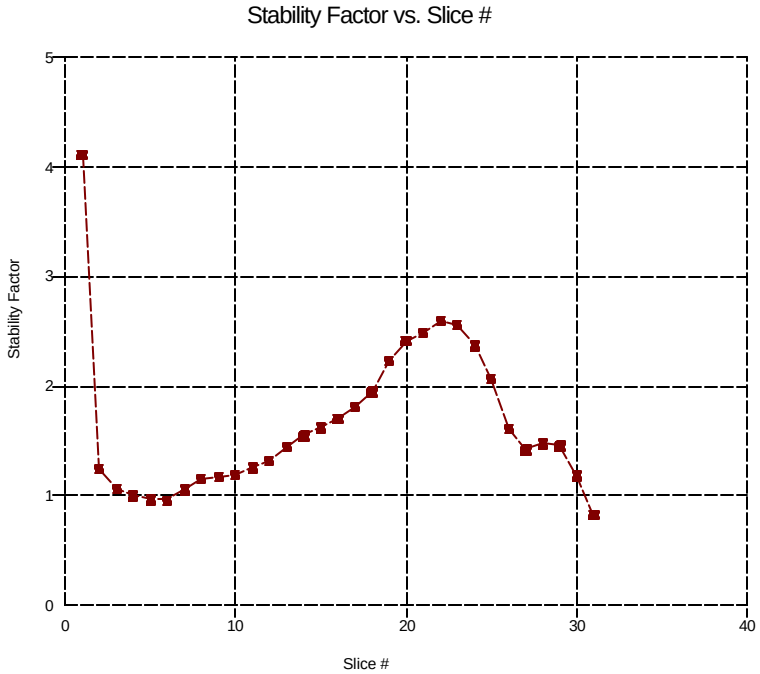


Figure 9.23 Local Stability Factor versus Slice #

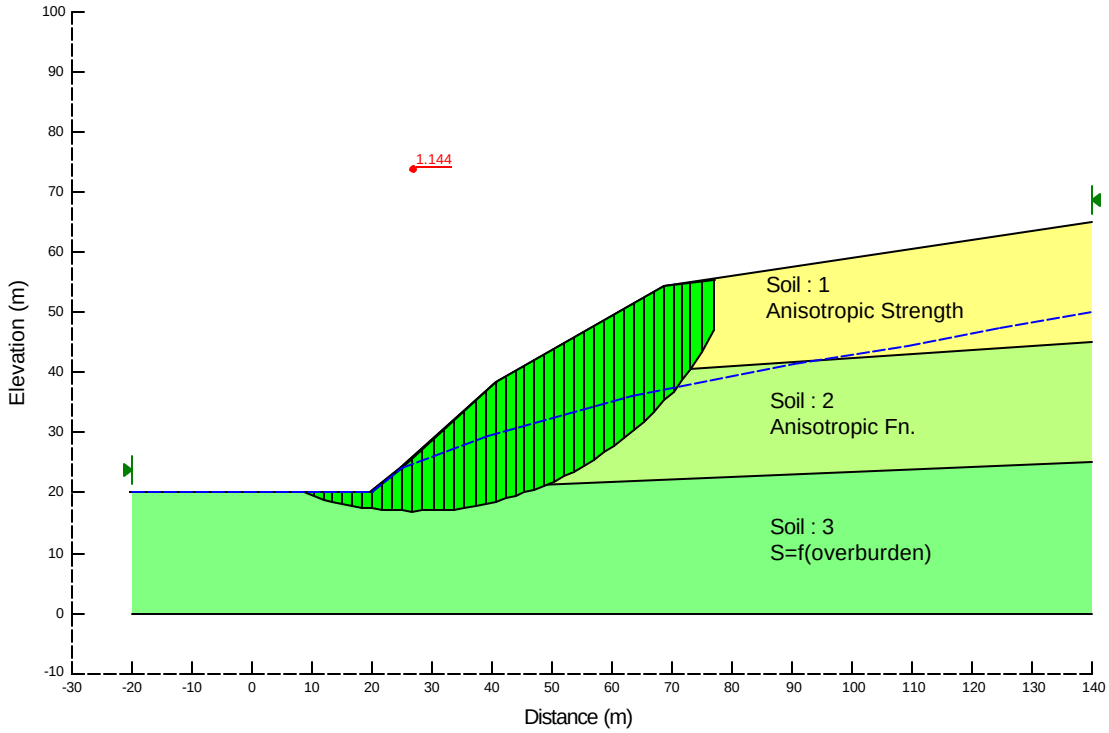


The SIGMA/W stress analysis was done by a simple "gravity turn on" procedure. Based on the finite element stresses, the stability factor is 1.412. The corresponding factor of safety is 1.428 when solved by the Morgenstern-Price limit equilibrium method.

Example with Anisotropic Strength

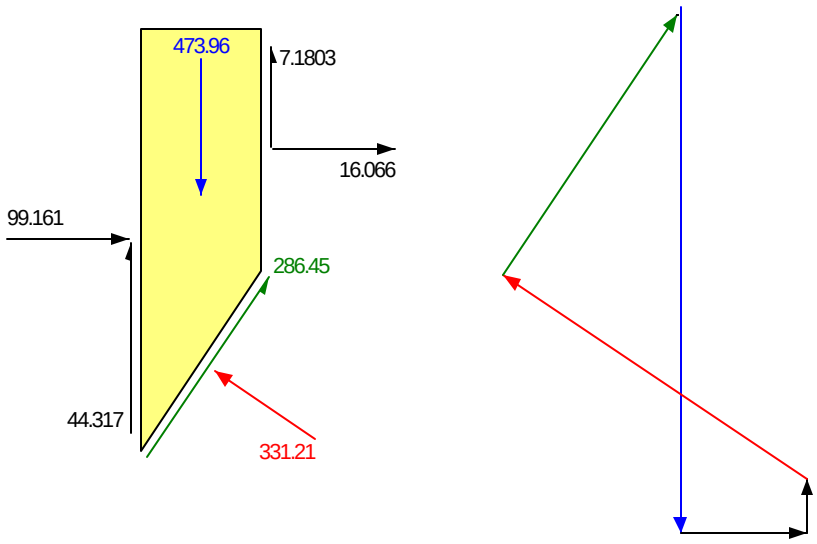
- File Name: ANISOTRO
- Primary Purpose: To show how anisotropic strength can be simulated in an analysis.
- Analysis Method: Spencer method.
- Special features: A single circular slip surface, anisotropic soils, tension crack simulated with a limiting angle on the slip surface

Figure 9.24 Example 11 - ANISOTRO



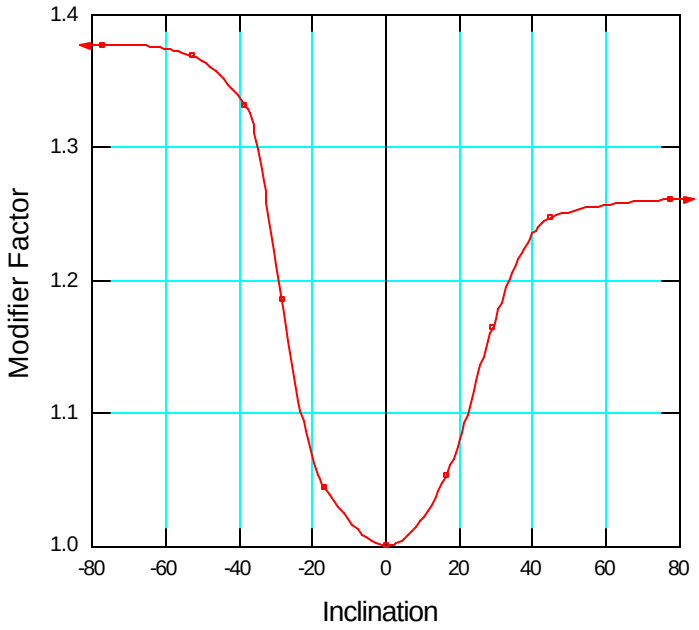
Soil 1 uses an anisotropic strength model in which the strength parameters c and f in both the horizontal and vertical directions are input. Considering slice 40, the base inclination angle is 56.11. The input c value is 20 in the horizontal direction and 25 in the vertical direction. The input f value is 30 in the horizontal direction and 35 in the vertical direction. Based on the inclination angle, the c and f values at the base of each slice are adjusted according to Equation 8.32 (see the Anisotropic Strength section in Chapter 8). After adjusting for anisotropy, the c and f values used in the shear strength calculation become 23.445 and 33.445 respectively. The mobilized shear force at the base of the slice can be calculated to be 286.45 which is consistent with the SLOPE/W result as shown in Figure 9.25.

Figure 9.25 Free Body Diagram and Force Polygon of Slice 40



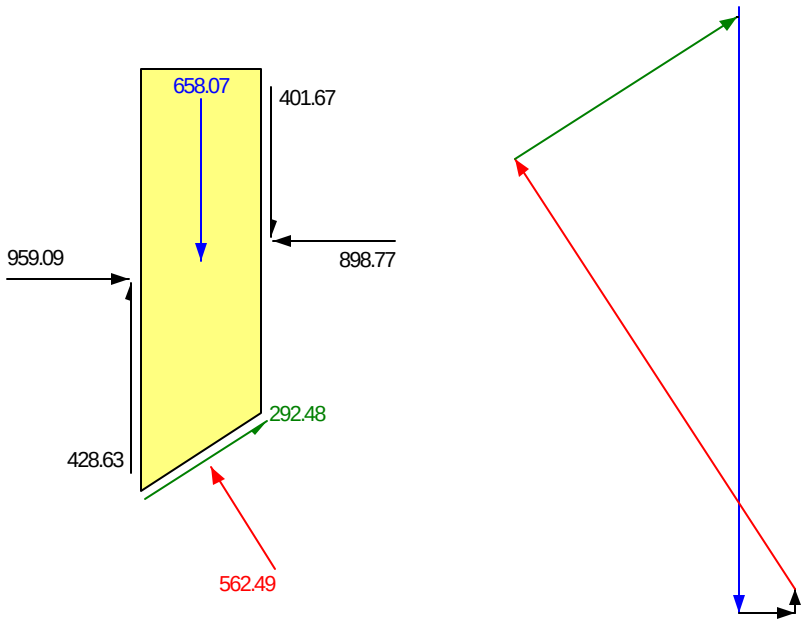
Soil 2 uses an anisotropic function model in which both the strength parameters c and f are modified according to a specified anisotopic function (Figure 9.26). This anisotropic function indicates that the input c and f values for horizontal slice base. When the base angle is not zero, the input c and f values must be multiplied by the modifier factor.

Figure 9.26 Anisotropic Function Used in the Example



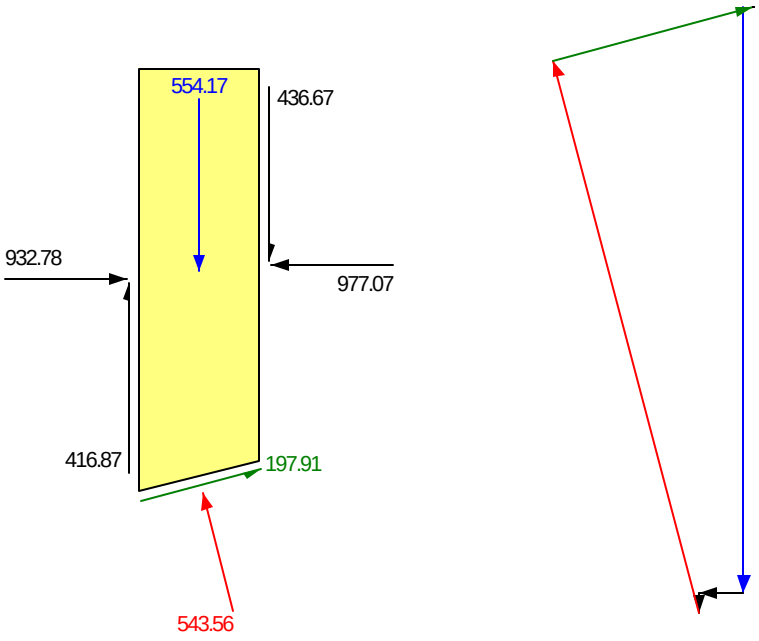
Considering slice 30, the base inclination angle is 32.932 and the input c and ϕ values for soil 1 is 20 and 30 respectively. The modifier factor for this slice is 1.194. Therefore, the c and ϕ values used in the shear strength calculation become 23.88 and 35.82 respectively. With the adjusted c and ϕ values, the mobilized shear force at the base of the slice can be calculated to be 292.48, which is the same as the SLOPE/W result, as shown in Figure 9.27.

Figure 9.27 Free Body Diagram and Force Polygon of Slice 30



Soil 3 uses the new model that estimate the shear strength as a function of effective overburden at the base of a slice. To account for anisotropy, the shear at the base of a slice is adjusted according to the specified anisotropic function (Figure 9.26). Considering Slice 20, the base inclination angle is 14.815 and the input Tau/Sigma Ratio is 0.55. The mobilized shear resistant force is 189 when computed from the effective overburden. From the anisotropic function, the modifier factor for this slice is 1.0439. Therefore, the mobilized shear resistant force is 197.91 after adjusting for anisotropy, as indicated on Figure 9.28.

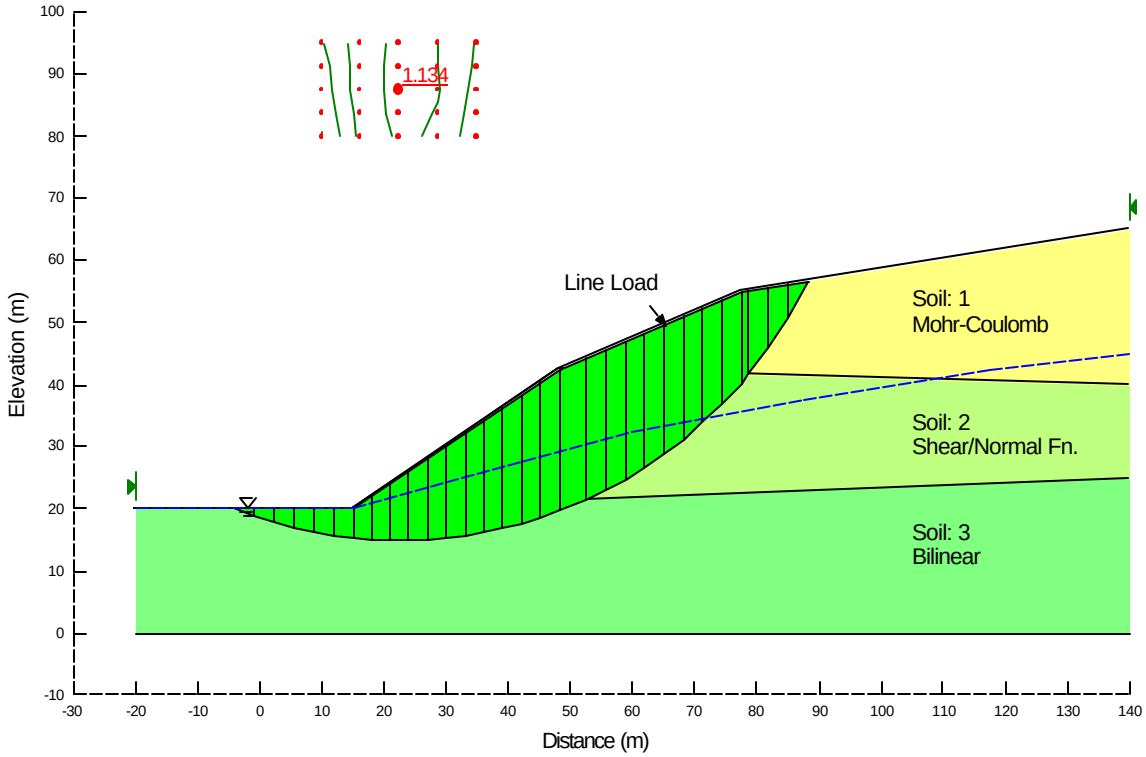
Figure 9.28 Free Body Diagram and Force Polygon of Slice 20



Example with Probabilistic Analysis

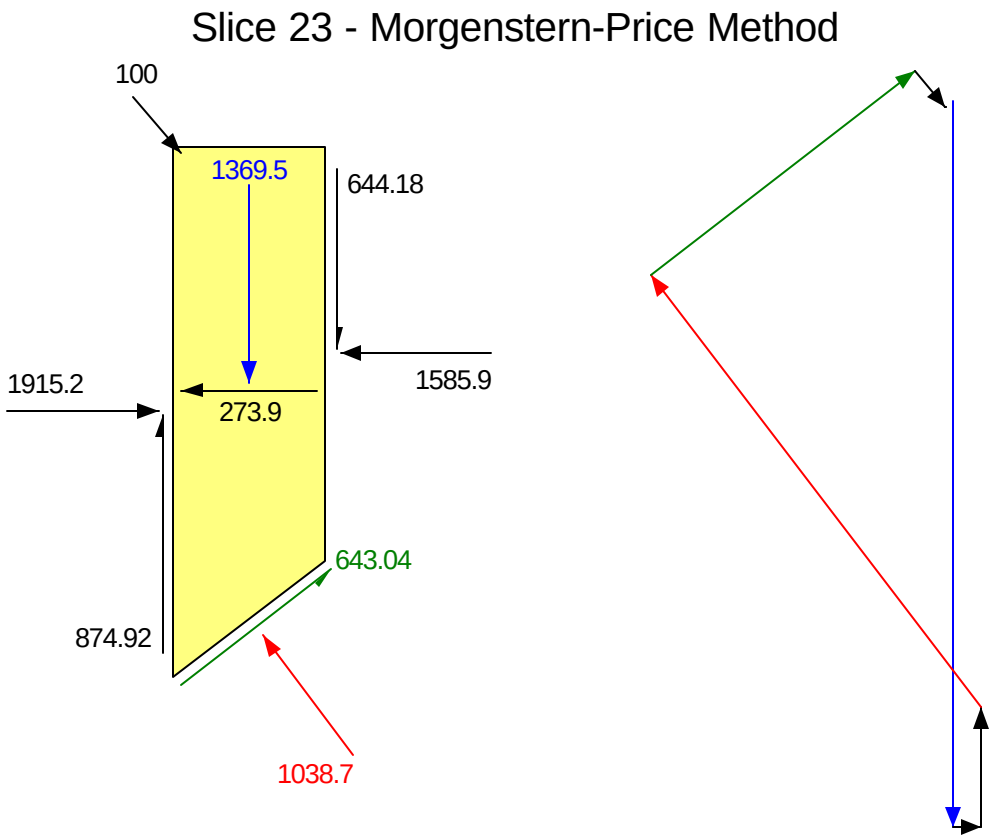
- File Name: PROBABI
- Primary Purpose: To show how a probabilistic analysis can be simulated.
- Analysis Method: Morgenstern-Price Method.
- Special features: Multiple circular slip surfaces, variability in shear strength parameters, pore-water pressure condition, line load and seismic load.

Figure 9.29 Example 12 - PROBABI



The free body diagrams and force polygons for Slice 23 is shown in Figure 9.30. The line load applied on the crest of the slope is specified as 100 kN, which is shown correctly on the free body diagram. The mean horizontal seismic coefficient is specified as 0.2 which corresponds to a seismic force of 273.9 acting horizontally inside the slice exactly as shown in the free body diagram.

Figure 9.30 Free Body Diagram and Force Polygon of Slice 23



The shear strength of Soil 2 is simulated with a Shear/Normal function in Figure 9.31. Based on the normal force and the pore water force of Slice 23 the effective normal stress on the base of the slice can be computed to be 227.9, which produces a shear stress of 186 from the specified Shear/Normal function. Using the base length of the slice and factor of safety, the mobilized shear force at the base can be calculated to be 643, which is identical to the mobilized shear force shown on the free body diagram.

Figure 9.31 Shear/Normal Function used for Soil 2

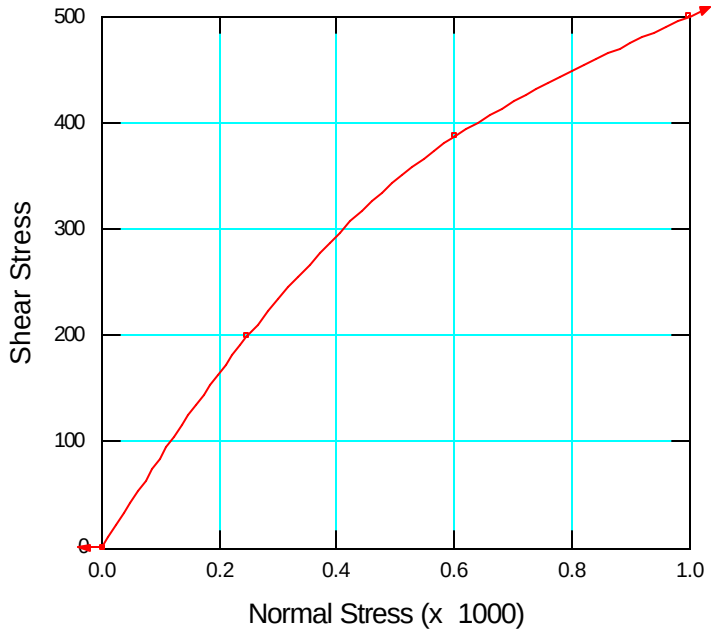


Figure 9.32 illustrates the relative frequency of the factor of safety (Probability Density Function) of the problem when variability of the parameters is considered. The histogram distribution is obtained by sorting and grouping all the factors of safety obtained in the Monte Carlo trials; the curve is the theoretical normal curve when the mean and standard deviation of the factor of safety are obtained. Since a normal distribution is assumed for the parameter variability, the computed factors of safety are also expected to be normally distributed. Figure 9.32 shows that the histogram distribution matches the normal curve reasonably well.

Figure 9.32 Probability Density Function of the Example PROBABI

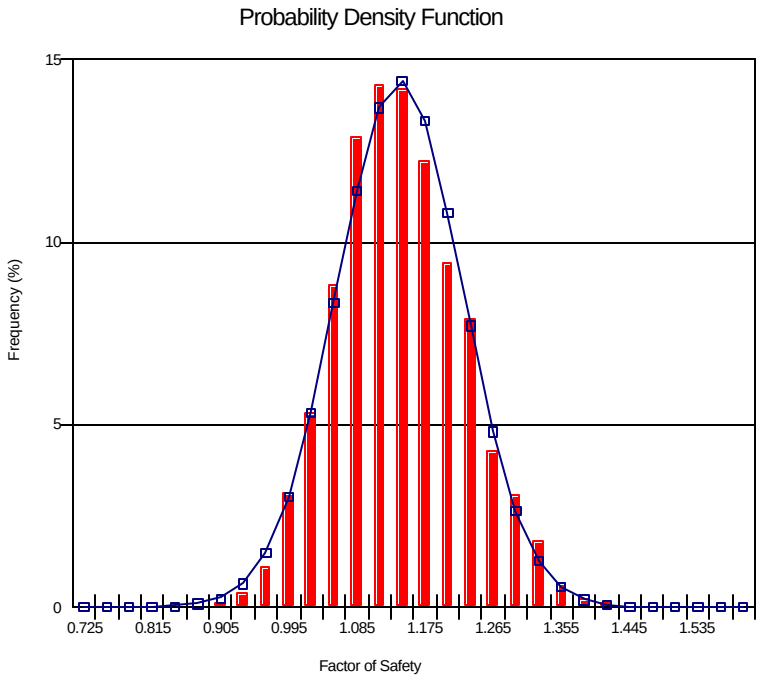
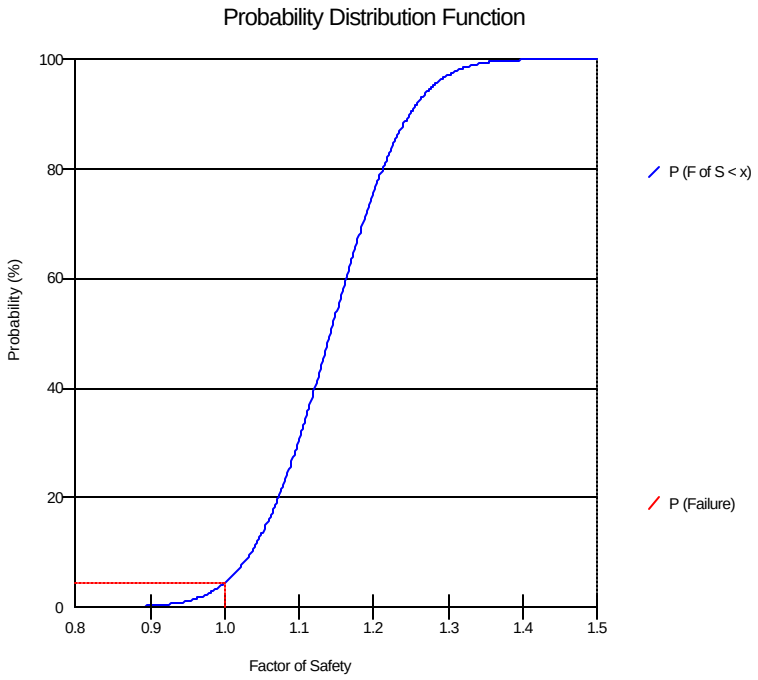


Figure 9.33 illustrates the probability of obtaining a factor of safety smaller than any specific factor of safety (Probability Distribution Function). The probability of failure, as shown by the red dotted line, indicates that there is a 4.25% chance that the slope will have a factor of safety of less than 1.0. The reliability index of this example problem is found to be 1.721. Based on the probability of failure versus reliability index chart produced by Christian, Ladd and Baecher, (1994) in Figure 8.25, the probability of failure is about 4.25%, which is consistent with SLOPE/W's solution (see Probability of Failure and Reliability Index in Chapter 8).

Figure 9.33 Probability Distribution Function of the Example PROBABI



The effect of the variability of the input parameters is also studied in this example, and the results using the Morgenstern-Price method are tabulated in Table 9.9. In all cases, the number of Monte-Carlo trials is set to be 5000. The presented results include the mean factor of safety, the reliability index and the probability of failure.

Table 9.9 Sensitivity of Probabilistic Results to Variation in Input Parameters

Parameters	Mean F of S	Reliability Index	Probability of Failure
Base Case - PROBABI	1.142	1.721	4.25%
SD-PWP decreased from 2.0 to 1.0	1.140	2.198	1.39%
SD-Horizontal Seismic Coeff decreased from 0.02 to 0	1.140	2.755	0.29%
SD-Phi (Soil 3) decreased from 3.0 to 0	1.139	3.787	0.01%
SD-C (Soil 3) increased from 2.0 to 5.0	1.139	3.146	0.08%
SD-Unit Weight (Soil 1) increased from 1.0 to 3.0	1.139	2.753	0.29%
SD-Phi (Soil 1) increased from 5.0 to 10.0	1.140	2.728	0.32%
SD-C (Soil 1) increased from 3.0 to 10.0	1.141	2.702	0.34%
SD-PWP increased from 1.0 to 2.0	1.144	1.909	2.80%
C-Phi Correlation Coefficient (Soil 1) changed from -0.5 to 0	1.141	1.899	2.87%

Starting from the base case of the PROBABI example, the standard deviations of several input parameters were decreased and then increased. As illustrated in Table 9.9, the mean factor of safety is not very sensitive to the parameter variations. In all cases, the mean factor of safety ranges from 1.139 to 1.144. Both the reliability index and the probability of failure are quite sensitive to the amount of variability in the input parameters. As anticipated,

when the amount of variability increases, the reliability index decreases, and the probability of failure increases. The probabilistic results are relatively insensitive to the C-Phi Correlation Coefficients.

See the Probabilistic Analysis section in Chapter 7 and the Probabilistic Slope Stability Analysis section in Chapter 8 for further discussion on how SLOPE/W performs probabilistic analyses.

References

Bishop, A.W. and Morgenstern, N., 1960. *Stability coefficients for earth slopes*. Geotechnique, Vol. 10, No. 4, pp. 164-169.

Box, G.E.P. and Muller, M.E., 1958. A note on the generation of random normal deviates. The Annals of Mathematical Statistics, American Statistical Association, USA., Vol. 29, pp.610-613.

Christian, J.T., Ladd, C.C. and Baecher, G.B., 1994. *Reliability Applied to Slope Stability Analysis*. Journal of Geotechnical Engineering, Vol. 120, No. 12. Pp. 2180-2207.

Fan, K., Fredlund, D.G. and Wilson, G.W., 1986. *An Interslice Force Function for Limit Equilibrium Slope Stability Analysis*. Canadian Geotechnical Journal, Vol. 23, No. 3, pp. 287-296.

Fellenius, W., 1936. *Calculation of the Stability of Earth Dams*. Proceedings of the Second Congress of Large Dams, Vol. 4, pp. 445-463.

Fredlund, D.G., 1974. *Slope Stability Analysis*. User's Manual CD-4, Department of Civil Engineering, University of Saskatchewan, Saskatoon, Canada.

Fredlund, D.G., and Krahn, J., 1977. *Comparison of slope stability methods of analysis*. Canadian Geotechnical Journal, Vol. 14, No. 3, pp. 429-439.

Fredlund, D.G., Zhang, Z.M. and Lam, L., 1992. *Effect of the Axis of Moment Equlibrium in Slope Stability Analysis*. Canadian Geotechnical Journal, Vol. 29, No. 3.

Grivas, D.A., 1981. *How Reliable are the Present Slope Failure Prediction Methods?* Proceedings of the Tenth International Conference of Soil Mechanics and Foundation Engineering, Stockholm, Sweden, Vol. 3, pp.427-430.

Harr, M.E., 1987. *Reliability-Based Design in Civil Engineering*. McGraw-Hill Book Company. pp. 290.

Higdon A., Ohlsen, E.H., Stiles, W.B., Weese, J.A. and Riley, W.F., 1978. Mechanics of Materials. John Wiley & Sons. pp.752. Janbu, N., Bjerrum, J. and Kjaernsli, B., 1956. *Stabilitetsberegning for Fyllinger Skjaeringer og Naturlige Skraninger*. Norwegian Geotechnical Publications, No. 16, Oslo.

Krahn, J., Price, V.E., and Morgenstern, N.R., 1971. *Slope Stability Computer Program for Morgenstern-Price Method of Analysis*. User's Manual No. 14, University of Alberta, Edmonton, Canada.

Lambe, T.W. and Whitman, R.V., 1969. *Soil Mechanics*. John Wiley and Sons, pp. 359-365.

Lapin, L.L., 1983. *Probability and Statistics for Modern Engineering*. PWS Publishers. pp. 624.

Li, K.S. and Lumb, P., 1987. *Probabilistic Design of Slopes*. Canadian Geotechnical Journal, Vol. 24, No. 4, pp. 520-535.

Lumb, P., 1966. *The Variability of Natural Soils.* . Canadian Geotechnical Journal, Vol. 3, No. 2, pp. 74-97.

Lumb, P., 1970. *Safety Factors and the Probability Distribution of Soil Strength* . Canadian Geotechnical Journal, Vol. 7, No. 3, pp. 225-242.

Morgenstern, N.R., and Price, V.E., 1965. *The Analysis of the Stability of General Slip Surfaces*. Geotechnique, Vol. 15, pp. 79-93.

Mostyn, G.R. and Li, K.S., 1993. *Probabilistic Slope Stability Analysis - State-of-Play*, Proceedings of the Conference on Probabilistic Methods in Geotechnical Engineering, Canberra, Australia. pp. 281-290.

Tan, C.P. Donald, I.B. and Melchers, R.E. , 1993. *Probabilistic Slope Stability Analysis - State-of-Play*, Proceedings of the Conference on Probabilistic Methods in Geotechnical Engineering, Canberra, Australia. pp. 89-110.

Whitman, R.V. and Bailey, W.A., 1967. *Use of Computer for Slope Stability Analysis*. Journal of the Soil Mechanics and Foundation Division of ASCE, Vol. 93, No. SM4.

Wolff, T.F, 1985. Analysis and Design of Embankment Dams: A Probabilistic Approach. Ph.D. Thesis, Purdue University, West Lafayette, IN. Yang, D., Fredlund, D.G. and Stolte, W.J., 1993. *A Probabilistic Slope Stability Analysis Using Deterministic Computer Software*, Proceedings of the Conference on Probabilistic Methods in Geotechnical Engineering, Canberra, Australia. pp. 267-274.

Index

—A—

Analysis control, 4-53
Analysis method, 4-44
Analysis options
  convergence, 4-53
  direction of movement, 4-53
  pore-water pressure, 4-54
  probability, 4-53
  slip surface, 4-54
  tension crack, 4-56
Anchor loads
  defining, 4-111, 4-149
  example, 9-25
Anisotropic function soil model, 4-65
Anisotropic Strength, 9-29
Anisotropic strength soil model, 4-64
Axes
  settings, 4-31
  sketch, 4-165

—B—

Bilinear soil model, 4-62
Bishop analysis method, 4-44
Block search, 9-19

—C—

Clipboard, 4-25
Color of soils, 4-37
Combined, S=f(datum) soil model, 4-66
Combined, S=f(depth) soil model, 4-66
Contours of pore pressure, 7-23
Convergence, 4-53
  effect of m-alpha, 8-13
  effect of steep slip surfaces, 7-11
  effect of weak layer, 7-12
Copy
  selection, 4-6
  to clipboard, 4-25
Corps of Engineers #1 analysis method, 4-47, 8-18
Corps of Engineers #2 analysis method, 4-48, 8-18
Critical slip surface position
  effect of soil properties, 7-10

—D—

Default settings, 4-23
Defaults
  font, 4-37
Draw
  anchor loads, 4-149
  contour, 6-24
  contour labels, 6-25
  graph, 6-29
  line loads, 4-146
  lines, 4-120
  menu, 4-118, 6-24
  points, 4-119
  points on mesh, 4-119
  pore-water pressure, 4-141
  pressure lines, 4-152
  probability, 6-36
  slip surface, 6-26
  slip surface axis, 4-129
  slip surface grid, 4-123
  slip surface left block, 4-132
  slip surface limits, 4-141
  slip surface radius, 4-126
  slip surface right block, 4-137
  slip surface specified, 4-130
  tension crack line, 4-155

—E—

Earth pressures, 7-18
Edit
  menu, 4-25, 6-12
Error messages
  factor of safety error codes, 6-29
Export drawing as a metafile, 4-19

—F—

Factor of safety
  choosing a method, 7-7
  definition, 8-4
  effect of interslice forces, 7-7
  effect of unsaturated soil, 8-23
  force equilibrium, 8-11
  invalid factors of safety, 6-29
  iterative procedures, 8-25
  moment equilibrium, 8-10
File
  exit, 4-13
  export, 4-19
  import data file, 4-16
  import picture, 4-17
  menu, 4-13, 6-9
  most recently used, 4-13, 6-9
  MRU file, 4-13
  new, 4-13, 5-3
  open, 4-15, 6-10

open SOLVE data file, 5-3
print, 4-22
save, 4-13
save as, 4-20
save default settings, 4-23
SL2 file, 4-15, 4-16
SL3 file, 6-10
Files
createdbySOLVE, 5-10, 5-17
factor of safety (FAC) file, 5-10, 5-17
probability (PRO) file, 5-15, 5-20
slice forces (FRC) file, 5-12, 5-17
Finite Element Stress analysis method, 4-44
Finite element stress method, 7-24, 8-44
Base center stresses, 8-46
base normal and shear, 8-47
example, 9-27
normal and mobilized shear, 8-45
stability factor, 8-44
Floating point precision, 7-6
Fonts
changing, 4-38
changing size, 4-37
default, 4-37
general, 6-21
sizes, 4-37, 6-20
Force polygon, 6-15
Free body diagram, 6-15

—G—

General limit equilibrium
theory, 8-9
General Limit Equilibrium (GLE) analysis method, 4-46, 8-9
Geofabric reinforcement, 7-16, 9-23
Graph
changing the display, 6-32
displaying a graph, 6-30
extracting the data, 6-34
Grid, 4-10
Grid of pore-water pressure head, 4-108, 4-145
Grid of pore-water pressures, 4-108, 4-145
Grid of Ru coefficients, 4-108, 4-145
Grid spacing, 4-29

—H—

Help
menu, 4-180, 6-43
on-line, 4-180
product support, 1-31

—I—

Identification of project, 4-42
Import
bitmap, 4-17
data file, 4-16
mesh, 4-16
metafile, 4-17
picture, 4-17
Installing
Evaluation Software, 2-9
SLOPE/W, 2-10
Interslice Force Functions, 4-48
Interslice forces, 8-16
Corps of Engineers, 8-18
Fredlund-Wilson-Fan, 8-20
Lowe-Karafiath, 8-19
IPX/SPX network protocol, 2-17, 2-18, 2-19, 2-21, 2-23, 2-27, 2-30, 2-45, 2-46, 2-47

—J—

Janbu analysis method, 4-44

—K—

KeyIn
analysis control, 4-53
analysis method, 4-44
anchor loads, 4-111
line loads, 4-110
lines, 4-85
menu, 4-41
points, 4-83
pore-air pressure, 4-109
pore-water pressure, 4-102
pressure lines, 4-115
project ID, 4-42
seismic load, 4-114
Shear/Normal strength function, 4-67, 4-79
slip surface axis, 4-93
slip surface grid & radius, 4-89
slip surface left block, 4-96
slip surface limits, 4-101
slip surface right block, 4-99
slip surface specified, 4-94
soil properties, 4-56
tension crack, 4-80

—L—

Labels
contour, 6-25
Lambda values, 7-22
Line loads
defining, 4-110, 4-146
Lines
defining for each soil, 4-85, 4-120
Loads

defining anchor loads, 4-111, 4-149
defining line loads, 4-110, 4-146
defining seismic load, 4-114
Lowe-Karafiath analysis method, 4-48, 8-19

—M—

Method of analysis, 7-7
Method of analysis, 4-44, 8-30
Modelling procedure, 7-4
Modify
menu, 4-166, 6-42
objects, 4-166
pictures, 4-171
text, 4-169
Mohr-Coulomb soil model, 4-60
Moment axis, 8-39
Morgenstern-Price analysis method, 4-45
Most Recently Used File, 6-9
Movement direction, 7-22
MRU File, 6-9

—N—

NetBIOS network protocol, 2-17, 2-18, 2-19, 2-23, 2-25, 2-27, 2-30, 2-41, 2-42, 2-43, 2-44, 2-46
Network Version
installing the Rainbow network software, 2-17
requirements, 2-17
running on Banyan Vines, 2-41
running on IBM LAN Server, 2-42
running on LANTastic, 2-43
running on Microsoft LAN Server, 2-44
running on Novell NetWare, 2-45, 2-46, 2-47
running on Windows 95, 2-47
running on Windows for Workgroups, 2-45
running on Windows NT, 2-46
security monitor program, 2-16, 2-17, 2-36
security server program, 2-13, 2-16, 2-17, 2-18
No Strength soil model, 4-61
Normal Force, 8-12

—O—

Ordinary analysis method, 4-44

—P—

Pictures
modify, 4-171
Piezometric Lines, 4-103, 4-142
Point information, 4-33
Points
defining, 4-83, 4-119
defining on a finite element mesh, 4-119
Pore-air pressure
defining, 4-109
Pore-water pressure
contours, 7-23
data points, 8-34
defining, 4-102, 4-141
finite element results, 8-36
options, 4-54
use of data points, 9-20
use of SEEP/W results, 9-21
Pore-water pressure contours, 4-106, 4-144
Pressure lines
defining, 4-115, 4-152
Print
at engineering scale, 4-23
drawing, 4-22
problems, 4-23
selection, 4-5
Printer
page, 4-26
settings, 4-22
Probabilistic analysis
example, 9-33
Probability
correlation coefficient, 8-50
density function, 6-36
displaying probabilistic results, 6-37
distribution function, 6-36
estimation of input parameters, 8-50
example problem, 9-33
Monte Carlo method, 8-48
Monte Carlo trials, 8-54
normal distribution function, 8-49
parameter variability, 8-48
probabilistic analysis, 8-48
probability of failure, 8-53
random number generation, 8-49
reliability index, 8-53
specifying a probabilistic analysis, 4-53
statistical analysis, 8-51
Problem extents, 4-28
Product support, 1-31

—R—

Redraw, 4-40
Ru Coefficients, 4-102, 4-103, 4-141, 4-142

—S—

S=f(datum) soil model, 4-63
S=f(depth) soil model, 4-63
S=f(overburden) soil model, 4-67
Scale, 4-27
engineering, 4-28
Security key, 2-13

NetSentinel, 2-11, 2-13, 2-16, 2-17, 2-18, 2-21, 2-23, 2-27, 2-30, 2-31, 2-33, 2-38
Sentinel system driver, 2-13, 2-17
SentinelPro, 2-11, 2-12
SEEP/W-computed heads, 4-109, 4-146
Seismic load
application, 7-13
defining the seismic coefficient, 4-114
Set
axes, 4-31
engineering scale, 4-27
menu, 4-26, 6-13
page, 4-26
snap to grid, 4-29
working area, 4-26
zoom factor, 4-30
Settings
save, 4-23
Shear/Normal function soil model, 4-64
Shear/Normal strength function, 4-67
Shear/Normal strength function, 4-79
graph window toolbar, 4-76
SIGMA/W-computed pressures, 4-109, 4-146
Sketch
arcs, 4-158
axes, 4-165
circles, 4-158
lines, 4-157
menu, 4-157, 6-41
modify pictures, 4-171
modify text, 4-169
text, 4-159
SL2 file, 4-15, 6-11
SL3 file, 6-10
Slice forces
viewing, 6-15
Slices
base normal, 8-12
width, 8-37
Slip surface
block specified, 9-19
circular, 9-17
composite, 9-17
defining fully specified slip surfaces, 4-94, 4-130
defining grid of rotation centers, 4-89, 4-123
defining projection angles, 4-89
defining radius lines, 4-89, 4-126
defining the axis point, 4-93, 4-129
defining the left block, 4-96, 4-132
defining the limits, 4-101, 4-141
defining the right block, 4-99, 4-137
direction of movement, 4-53
displaying any slip surface, 6-26
displaying the minimum slip surface, 6-26
fully specified, 9-18
options, 4-54
projection angles, 7-11
use of projection, 9-23
Slip surfaces
axis, 8-39
Snap to Grid, 4-10
Soil Models
Anisotropic function), 4-65
Anisotropic strength, 4-64
Bilinear, 4-62
Combined, S=f(datum), 4-66
Combined, S=f(depth), 4-66
Mohr-Coulomb, 4-60
No strength, 4-61
S=f(datum), 4-63
S=f(depth), 4-63
S=f(overburden), 4-67
Shear/Normal function, 4-64
Undrained, 4-61
Soil properties, 4-56
Soil strength models, 8-41
anisotropic strength, 8-41
anisotropic strength function, 8-42
strength function, 8-42
Spencer analysis method, 4-45
Spline interpolation, 8-34
Stability factor, 8-44
definition, 8-44
Status Bar, 4-39, 6-22
viewing, 4-39, 6-22
Structural elements, 7-17
Submergence
complete, 7-21
partial, 7-20
Surface pressure
defining, 4-115, 4-152

—T—

TCP/IP network protocol, 2-17, 2-18, 2-19, 2-23, 2-26, 2-27, 2-29, 2-48
Tension crack
defining, 4-80, 4-155
effect on convergence, 7-11
Tension crack options, 4-56
Text
font sizes, 4-37
modify, 4-169
Toolbars, 4-4, 6-4
grid, 4-10
method, 6-7
mode, 4-7, 6-5

standard, 4-4, 6-4
view preferences, 4-9, 6-6
viewing, 4-39, 6-22
zoom, 4-11
Tools
CONTOUR, 4-179
menu, 4-176
SOLVE, 4-179
verify, 4-176

—U—

Undrained soil model, 4-61
Units
consistent set, 7-5
engineering units, 4-27
weight of water, 4-29
Unsaturated shear strength, 8-24

—V—

Verification
with hand calculations, 9-4
with infinite slope case, 9-12
with other programs, 9-15
with stability charts, 9-10
Verify DEFINE data, 4-176
View
grid, 4-11
menu, 4-33, 6-14
method, 6-14
point information, 4-33
preferences, 4-36, 6-18
redraw, 4-40
slice forces, 6-15
toolbars, 4-39, 6-22

—W—

Working area, 4-26

—Z—

Zoom
control, 4-11
factor, 4-30
in, 4-11
objects, 4-11
out, 4-11
page, 4-11